Copy No. 12

DEFENCE R&D DÉFENSE

DEVELOPMENT OF THE ISSMM SOFTWARE TOOL FOR RESIDUAL STRENGTH ASSESSMENT OF SHIP STRUCTURES

N. Pegg

DEFENCE RESEARCH ESTABLISHMENT ATLANTIC

Technical Memorandum

DREA TM 1999-055

March 1999

National Defence Défense nationale

Canada

102-02839/

National Defence
Research and Development Branch

Défense nationale
Bureau de recherche et développement

DREA TM 1999/055

Development of the ISSMM Software Tool For Residual Strength Assessment of Ship Structures

N. Pegg

DEFENCE RESEARCH ESTABLISHMENT ATLANTIC
PO Box 1012, Dartmouth
Nova Scotia, Canada
B2Y 3Z7

Approved by: J.L. Kennedy
Head, Warship Signatures & Safety Section

March 1999

TECHNICAL MEMORANDUM

Prepared by

Defence Research Establishment Atlantic

Centre de Recherches pour la Défense Atlantique

Canada

Abstract

This technical memorandum summarizes plans and information used in preparing the prime contract specification for the development of a software tool for the Defence Research and Development Board (DRDB) major development project Improved Ship Structural Maintenance Management (ISSMM). It contains general information and descriptions of existing capabilities under the following headings: development environment; Halifax Class geometric, structural and material databases; operational sea loads; global/detail finite element analysis; damage modelling and analysis; ultimate strength assessment; fatigue crack initiation and growth assessment; component stress and stability assessment; and mission risk assessment. This document is intended to provide optional guidance to the contract bidders in preparing their proposals. It also serves as a discussion of the state of the art in DND for ship structural analysis.

Resume

Le présent document technique fait la synthèse des plans et des renseignements utilisés pour préparer le devis du contrat initial en vue de l'élaboration d'un logiciel-outil pour le projet de développement majeur de la Direction de la recherche et du développemetn pour la défense intitulé Gestion améliorée de la maintenance de la structure des navires (ISSMM). Il contient des renseignements généraux ainsi que la description des installations existantes sous les rubriques suivantes : environnement de développement; bases de données géométriques et structurales et base de données du matériel de la classe Halifax; charges de mer opérationnelles; analyse globale et détaillée des éléments finis; modélisation et analyse des dommages; évaluation de la limite de rupture; création de fissures par fatigue et évaluation de l'accroissement des fissures; évaluation de la contrainte sur les composants et de la stabilité des composants; évaluation des risques de la mission. Le présent document vise à aider les soumissionnaires, au besoin, à préparer leurs soumissions. Il sert aussi de description de l'état actuel des réalisations du MDN en matière d'analyse structurale des navires.

Table of Contents

Abstract 2
Resume 2
Table of Contents 3
EXECUTIVE SUMMARY 5
Introduction 5
Principal Results 5
Significance of Results 6
Future Plans 6
1. Introduction 7
2. Aim 7
3. Background 7
4. Development Environment 9
4.1. The ISSMM User 9
4.2. Hardware 9
4.3. Hierarchical Object Oriented Data Management 10
4.4. Legacy Functional Code 10
4.5. Object Library 11
4.6. Graphical User Interface (GUI) 11
4.7. Prototyping 12
4.8. Existing Tools 12
5. Halifax Class Database 13
5.1. Relational Hierarchical Object-Oriented Database 13
5.2. Geometric Database 13
5.3. Structural Finite Element Database 15
5.4. Material Database 15
5.5. Existing Tools 16
6. Operational Sea Loads 17
6.1. Structural Load Requirements 17
6.1.1 Still Water Bending Moment Load Requirements 18
6.1.2 Hull Girder Ultimate Strength Load Requirements 18
6.1.3 Component Failure Load Requirements 18
6.1.4 Fatigue Load Requirements 19
6.2. Operational Profiles 19
6.3. Response Amplitude Operators (RAOs) 20
6.4. Nonlinear Sea Load Effects 20
6.5. Load Library 21
6.6. Existing Tools 21
7. Global/Detail Structural Analysis 23
7.1. Global Finite Element Model 23
7.2. Global Finite Element Analysis and Application of Loads 24
7.3. Smart Structural Detail Model Generation 24
7.4. Top-Down FEA 26
7.5. Model Checking and Result Display 28
7.6. Suggested Existing Tools 28

8. Damage Modelling and Analysis 29
8.1. Corrosion Modelling 30
8.2. Deformation Modelling 30
8.3. Crack Modelling 31
8.4. Assessing the Effects of Damage 31
8.5. Considerations for Modelling Gross (Weapons) Damage 32
8.6. Existing Tools 32
9. Ultimate Strength Assessment 33
9.1. Scope 33
9.2. Ultimate Strength Modelling Assumptions 34
9.3. Load-Shortening Curves 34
9.4. Integration of Damage Models 34
9.5. Existing Tools 35
10. Fatigue Crack Assessment 36
10.1. Scope 37
10.2. Crack Initiation 37
10.2.3 Nominal Stress Approach 38
10.2.3 Hot Spot Stress Approach 38
10.2.3 Local Notch-Tip Strain Approach 38
10.3. Crack Growth 39
10.4. Fatigue Calculations using Extreme Load 39
10.5. Fatigue Calculations using Operational Load Spectrum 40
10.6. Residential Strain Effects 40
10.7. Existing Tools 40
11. Structural Component Stress and Stability Assessment 41
11.1. Scope 42
11.2. Definition of Structural Components 42
11.3. Stress and Buckling Stability Limits 42
11.4. Existing Tools 43
12. Mission Risk Assessment 43
12.1. Probabilistic and Deterministic Risk Assessment Methods 43
12.2. Failure Limit States 44
12.3. Presentation of Risk to the Operator 44
13. Testing 45
13.1. Software Integrity Testing 45
13.2. Software Functionality Testing 45
14. Documentation 45
14.1. User's Manual 45
14.2. Programmer's Software Management Manual 46
14.3. On-line Help and Documentation 46
14.4. Theory and Functionality Manual 46
14.5. Tutorial 46
15. Existing ISSMM Contract Work 46
16. Concluding Remarks 47
17. Acknowledgements 47
18. Reference Documents 47

Development of the ISSMM Software Tool for Residual Strength Assessment of Ship Structures

Neil Pegg

EXECUTIVE SUMMARY

Introduction

A Defence Research and Development Board (DRDB) major project entitled 'Improved Ship Structural Maintenance Management (ISSMM)' is being developed to provide an efficient tool to aid in the assessment of the effect of various types of damage and degradation on the structural integrity and operational capability of Canadian Forces warships. The project was initiated in the early 1990's with concept definition studies undertaken in industry and in-house by Defence Research Establishment Atlantic (DREA). This initial phase was followed by proof of concept studies, the advancement of most of the application tools for specific types of structural and sea loads analyses and the initiation of material and structural databases. The initial developments of an integrated hierarchical object-oriented data management system for ship structures for ISSMM have also been undertaken by DREA to provide a framework for the overall software development and integration. The next step in the development of ISSMM is to produce the overall software package that will complete the development of, and integration of, the required analysis functions and their necessary databases. This technical memorandum summarizes development plans and information used in preparing the prime contract statement of work specification for the ISSMM software tool for the residual strength assessment of damaged CF vessels. The original ISSMM planning document, DREA TM 97/298, has been used extensively, and this memorandum serves as an update to that reference.

Principal Results

This technical memorandum contains general information and descriptions of existing capabilities under the general technology areas of development environment, Halifax Class geometric, structural and material databases, operational sea loads, global/detail finite element analysis, damage modelling and analysis, ultimate strength assessment, fatigue crack initiation and growth assessment, component stress and stability assessment, and mission risk assessment. During the initial project definition and concept proof phases of ISSMM development, there have been significant advances in all of these technology areas, both through in-house DRE work and industrial research contracts. This work is described and referenced such that potential bidders for the ISSMM prime contract can evaluate whether to adopt existing technology or propose alternatives. In addition to providing optional guidance to the contract bidders in preparing their proposals, this document also serves as a discussion of the state of the art in DND for ship structural analysis.

Significance of Results

The proof of concept phase of ISSMM has been very valuable in reducing the risk of applying various advanced technologies to the assessment of the effects of damage on the operational capability of CF vessels. The development of methods to predict operational sea loads and their validation through the HMCS NIPIGON sea trials, has provided an approach to developing realistic structural load cases which is a significant advancement over current industry practice. The top-down finite element modelling approach and structural evaluation codes developed for ISSMM are being used by DND, other Navies and international ship classification societies and have developed into a successfully marketed product for Canadian industry. The material-testing program undertaken for ISSMM has provided a basis for fatigue and fracture assessment of the Halifax Class. The hierarchical data management approach developed for ISSMM is a major advancement in developing engineering software and will provide a means to ensure a robust and manageable tool.

Future Plans

The development of the ISSMM software tool has been approved, and will likely begin in 1999 for a 3-year duration. DREA and the Directorate of Maritime Ship Support (DMSS-2) staff will be involved in producing this tool in conjunction with a prime contractor. Much of the technology described in this report will be part of the finished product. The ISSMM tool will be developed with consideration of future modifications such that improvements can be easily added to new versions of the software as the various related technologies advance.

1. Introduction

A Defence Research and Development Board (DRDB) major project entitled 'Improved Ship Structural Maintenance Management (ISSMM)' is being developed to provide an efficient tool to aid in the assessment of the effect of various types of damage and degradation on the structural integrity and operational capability of Canadian Forces Warships. The project was initiated in the early 1990's with concept definition studies undertaken by industry and in-house by DREA. This initial phase was followed by proof of concept studies, the advancement of most of the application tools for specific types of structural and sea loads analyses and the initiation of material and structural databases. The initial developments of an integrated hierarchical object-oriented data management system for ship structures for ISSMM have also been undertaken by DREA to provide a framework for the overall software development and integration. The next step in the development of ISSMM is to produce the overall software package that will complete the development of, and integration of, the required analysis functions and their necessary databases. This technical memorandum summarizes development plans and information used in preparing the prime contract statement of work specification for the ISSMM software tool for the residual strength assessment of damaged CF vessels. The original ISSMM planning document, reference [1], has been used extensively, and this memorandum serves as an update to that reference.

2. Aim

The aim of this document is to provide the background information from which the formal contract specification for the functional requirements of the ISSMM software tool was produced. The ISSMM tool is to be used for assessing the risk of operating structurally damaged or degraded CF warships. The document makes considerable reference to other related documents, some of which have been produced in the ISSMM concept development phase.

3. Background

The Canadian Navy initiated improvements to its maintenance practices for naval vessels through a program entitled the Naval Ship Maintenance Program (NSMP). The NSMP encompassed new survey and repair procedures and guidelines, a computer database to track inspection and repair actions (Structural Inspection Database - SID), and a process for certifying the structural integrity of each vessel (Ship Structural Integrity Program - SSIP). An inherent requirement of this overall program is the need to determine the structural capability of the vessel in its current condition to meet its operational requirements. To meet this need, DND initiated a major development project, ISSMM (Improved Ship Structural Maintenance Management), to develop an integrated computer-based approach using modern methods of analysis, to efficiently assess the change in structural integrity of a CF vessel due to damage or degradation. The ISSMM project was initiated in April 1996 after a three-year concept planning stage [1,2,3,4,5].

Classical rules-based ship structural design and analysis methods do not lend themselves to determination of the operational structural integrity of an in-service vessel in a degraded condition. There have been significant recent advances in ship structural analysis technology such as the finite element method, 3-D panel methods for sea loads,

ultimate strength programs, and fatigue and fracture assessment methods. These advances, along with the rapid advancements in computing technology, have made it possible to develop an integrated method to evaluate the structural capability of a vessel, including the effects of degradation. The goal of the ISSMM project is to make this technology readily and effectively accessible to the CF naval architecture and marine engineering community.

Through earlier phases of the ISSMM project, methods and concepts to be used for structural assessment have been partially or fully developed. The existing analysis methods that have been used in the concept development studies are a combination of in-house developments, commercial acquisitions (only a small component), contractor developments (including some where the contractor has contributed their own IP), and developments contributed by other nations. These methods are discussed in the following functionality descriptions as *existing tools*.

A flow chart showing the modules of the ISSMM program is given in Figure 1.

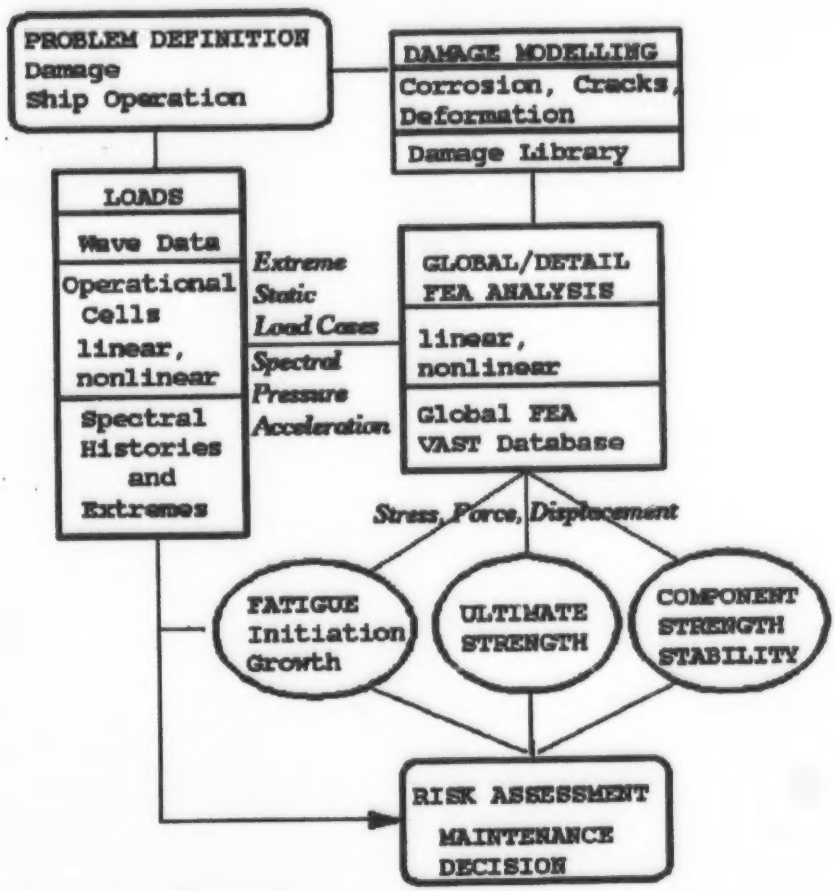

Figure 1: ISSMM Flowchart

The flowchart outlines the basic functionality of the ISSMM software. The process is initiated by defining the ship, the damage and the intended operation of the vessel. This information is used to progress to the definition of structural sea loads to be used in the assessment and to create structural models incorporating the damage. Finite element analysis is used to translate the structural sea loads into load effects for the damaged and undamaged vessel and these load effects are then used in the assessment of structural integrity through fatigue, ultimate strength, and component strength and structural stability failure limit states. These failure limit states are used to aid the operator in performing a risk assessment and in evaluating repair and operational alternatives.

4. Development Environment

This section describes the general computer hardware and software development considerations for ISSMM.

4.1. The ISSMM User

The software tool produced by the ISSMM project is to be used by Naval Architects and Engineers in the Directorate of Maritime Ship Support (DMSS) and DND staff residing in the coast dockyard support facilities. While the user is expected to be knowledgeable and competent in the field of ship structures and maintenance and repair strategies, the user is not expected to be experienced in applying finite element analysis, advanced sea load codes, fracture mechanics, or reliability methods to ship structures. The ISSMM software will look after all data access and formatting and the user should not be expected to be familiar with the data requirements of the individual application programs.

The ISSMM software tool is expected to act as an expert system to aid the user in applying advanced methods of analysis to the Halifax Class structure. While developing a tool which will have as wide a range of application as possible, it is recognized that an expert system approach will have a scope that is limited to those types of problems which are considered in its development. For problems beyond the scope of the ISSMM software, expertise in the advanced methods of analysis will be required and may be supplied by DREA or contractors.

4.2. Hardware

Initial development plans for ISSMM [1], which were undertaken in the early 1990's, envisioned the need for a high-level work station to meet the computing requirements. Since that time, developments in the PC field have made it practical to produce ISSMM for a PC.

The finite element meshes that have been created in the concept development stage are of a considerable size (100,000-plus degrees of freedom) and appreciable CPU and disk space will be required to efficiently manage the various ISSMM analyses. The current state of readily available PC hardware gives a reasonable minimum requirement for ISSMM of a 450 MHz CPU processor with a 10 Gigabyte hard drive and 128 Megabytes of RAM. It is expected that, within the delivery time frame of ISSMM, a PC of considerably greater capability will be readily available.

Specified maximum analysis times for a set of ISSMM sample problems will ultimately define the computer hardware and software efficiency requirements. Given

that ISSMM is supposed to be a rapid engineering assessment tool, any conceived ISSMM problem solution should take no more than a day from problem definition to a risk assessment result, with most problems taking less time.

It is also recommended that the same minimum hardware requirements be placed on the contractors who are developing the code to ensure efficient code production.

4.3. Hierarchical Object Oriented Data Management

Developing a software tool that will rapidly undertake a variety of complex analysis tasks on a ship structure is a challenge. Inherent in this process is the need to minimize user data input. The program must contain as complete a database as possible for the Halifax Class structure, connection types, materials, loads, operations, existing analysis models, etc. and be 'smart' in that the program will need to recognize when it needs data and be able to automatically access it.

Relational databases provide part of this function, but existing commercial relational databases will require considerable programming to make them effective for ship structural analysis.

Modern software development uses the object-oriented programming approach. Databases for ship structure using an object-oriented approach are a possible solution to the ISSMM data management requirements. Defining all ship structural components as objects which have specific attributes related to their structural configuration, finite element modelling, loading, materials and to what other structure they are attached, can result in an effective database tool.

The hierarchical component added to the object-oriented data manager allows a natural flow of data handling from larger to smaller structural components, starting with the ship, progressing to components such as decks and then stiffeners, plates and connections and, possibly, even welds. DREA has begun development of the HOOD (Hierarchical Object-Oriented Data manager) system for this purpose. It is discussed further below.

4.4. Legacy Functional Code

Most of the structural analysis and sea load codes which have been developed to date throughout the world have been written in the Fortran programming language and have not followed an object-oriented approach nor use any modern relational data management systems. It is not practical to reproduce these codes in an object-oriented manner, so it is necessary to provide interfaces (sometimes called wrappers) which allow the use of these codes in an object-oriented graphical user interface (GUI) and relational data management system.

In general, these codes require input data files that could be derived from the overall Halifax Class database and data management system. They produce output data files that can be used by the GUI and graphics programs to display results. Figure 2 shows the functional modules required for ISSMM. These are linked to a central GUI and data manager by input/output (I/O) gates. The names of legacy code that may be used for ISSMM are printed outside of the module boxes. These codes are further described and referenced in the following sections of this document.

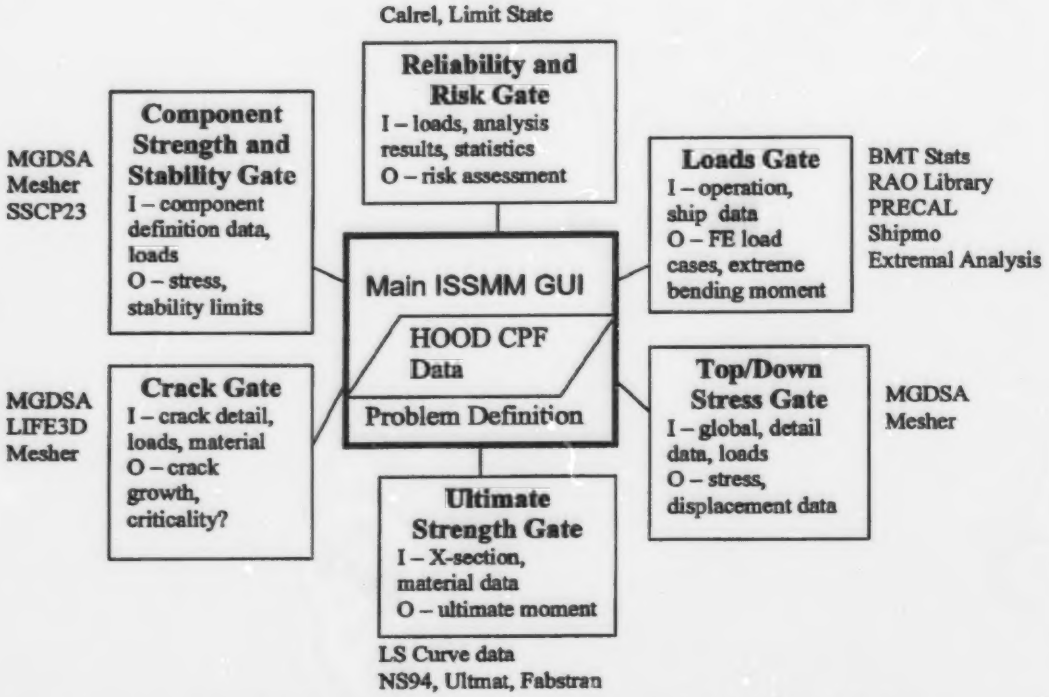

Figure 2: Schematic of ISSMM Operating Modules, Gates and Associated Software

4.5. Object Library

In order to produce a modern efficient program, the ISSMM software development will use object-oriented programming. The resultant code will be sufficiently large that a library of the programming objects should be developed including user documentation. The object library will provide a means to efficiently access the many programming objects that will be produced, possibly through an internet site, with the objective of promoting object 're-use' by all of the ISSMM development participants. Future-use of the objects for updating ISSMM and producing other similar codes, should also be considered in developing the library. This methodology follows a "Domain Engineering" approach to software development.

4.6. Graphical User Interface (GUI)

The ISSMM software tool should be developed using the Visual C++ and Microsoft Foundation Class (MFC) user interface environments as these are the most common tools used in developing modern interactive engineering software. Visual Basic and Component Object Modeling (COM) should also be investigated where their functionality may suit the application. The GUI should include pull-down menus, on-line

help, and 3-D graphics visualization, and allow the user to undertake the full ISSMM analysis within the operating shell. Online help capabilities should be written to support the non-expert ISSMM users. Examples of two existing user shells developed as part of the ISSMM concept proof stage are shown in Figure 3.

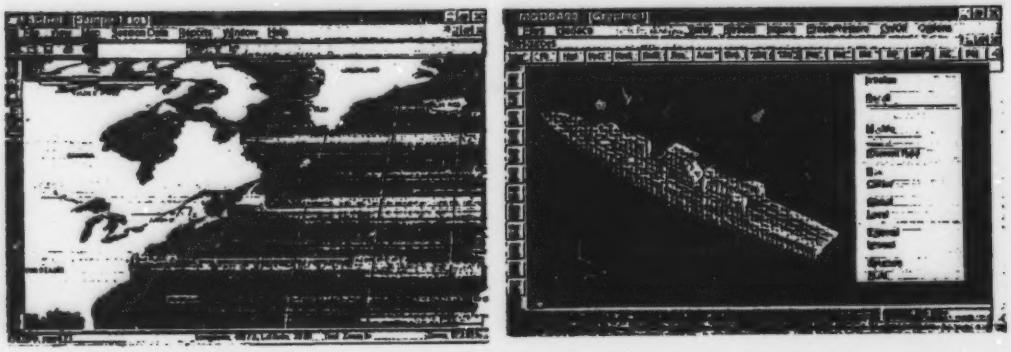

Figure 3: Existing Operating Shell GUIs for Sea Loads Definition and Structural Analysis

4.7. Prototyping

Developing an acceptable prototype early in the ISSMM development will save costly code modifications to satisfy user needs at the end of the project. The ISSMM software package should be initially developed as a prototype containing all of the functionality options and displaying the 'look and feel' of the program for the acceptance of potential users. The prototype should be reviewed by a number of DREA, Naval Engineering and Naval Architecture staff before proceeding with full development.

4.8. Existing Tools

HOOD: [6] DREA has been developing the Hierarchical Object-Oriented Data (HOOD) management system for ship structures and other ship modeling/simulation software applications. HOOD's primary role is to serve as a repository of data structure and ship related objects for reuse in developing new applications. HOOD objects can be used to transfer data between applications and modules of the ISSMM software development. The HOOD system is under development and will have to be expanded to meet the requirements of the ISSMM program. Figure 4 shows a schematic of the HOOD system capabilities, and Figure 5 shows a page of the existing HOOD documentation describing one object for managing beam data. The MARI STEP referred to in Figure 4 is an international data standard being developed (see related Web sites for details). While it is not mandatory to have ISSMM directly use the STEP modeling standard (mainly because it will not be fully developed or implemented in the ISSMM time frame), it is desirable to have links to and from STEP. HOOD is written in C++ and is owned by the Crown.

Visual C++: This programming environment has been used in developing ISSMM GUIs and some more recent application modules. Visual C++ is commercially available.

Microsoft Foundation Class: This set of objects has been used in developing ISSMM GUIs and some applications. It is commercially available.

Object Outline: This application has been used in creating HTML / Rich Text or Microsoft Help online document for HOOD objects. It is commercially available.

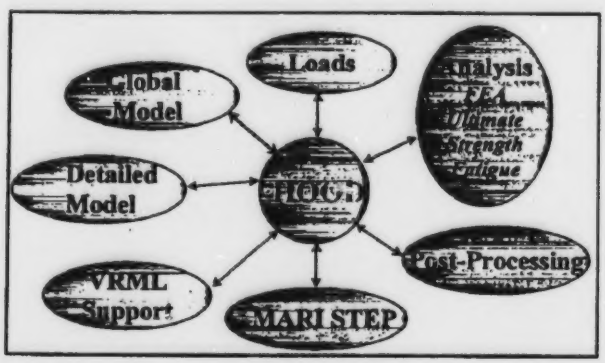

Figure 4: Schematic of HOOD Data Manager for Ship Structure

5. Halifax Class Database

An important feature of ISSMM is to have as much data as possible that is required for the structural assessment, readily available to the operator of the ISSMM software. This will be accomplished by developing an extensive, efficient database as part of the software. ISSMM is being developed initially to address the Halifax Class frigate. To meet future requirements, facilities to create databases for other types of vessels should be considered during the development of ISSMM. General particulars on the Halifax Class frigate can be found at the web sight, http://www.marlant.hlfx.dnd.ca/marlant/halifaxe.html.

5.1. Relational Hierarchical Object-Oriented Database

As discussed above, an effective means of providing data is through an object-oriented relational data management system. Each ship entity such as decks, lines of form, plating, stiffeners, etc. can be described as objects that have their own data managers. These data managers operate in a hierarchical manner to allow each entity to know what structure and functions are related to it. Material properties, loads, structural finite element models, etc. are all accessed or developed through the information stored and created by the data management system.

5.2. Geometric Database

The basic data structure will be a geometric database of the Halifax Class. This level will be managed by the various data management objects and contain all structural and materials information for all levels of the ship from decks and bulkheads down to individual structural detail connections. These graphical objects should be smart enough

to know what structure they are related to and automatically propagate changes between related objects. Graphical and numeric search capabilities will locate the area of interest on the structure. All of the necessary information to produce structural models or to find existing structural models will be identified by the program. The HOOD data management system provides some of the necessary functionality now and, should it be used for ISSMM, will have to be expanded to include all necessary data managers for ISSMM. Figure 6 shows a partial geometric database for the Halifax Class with some of the associated data managers.

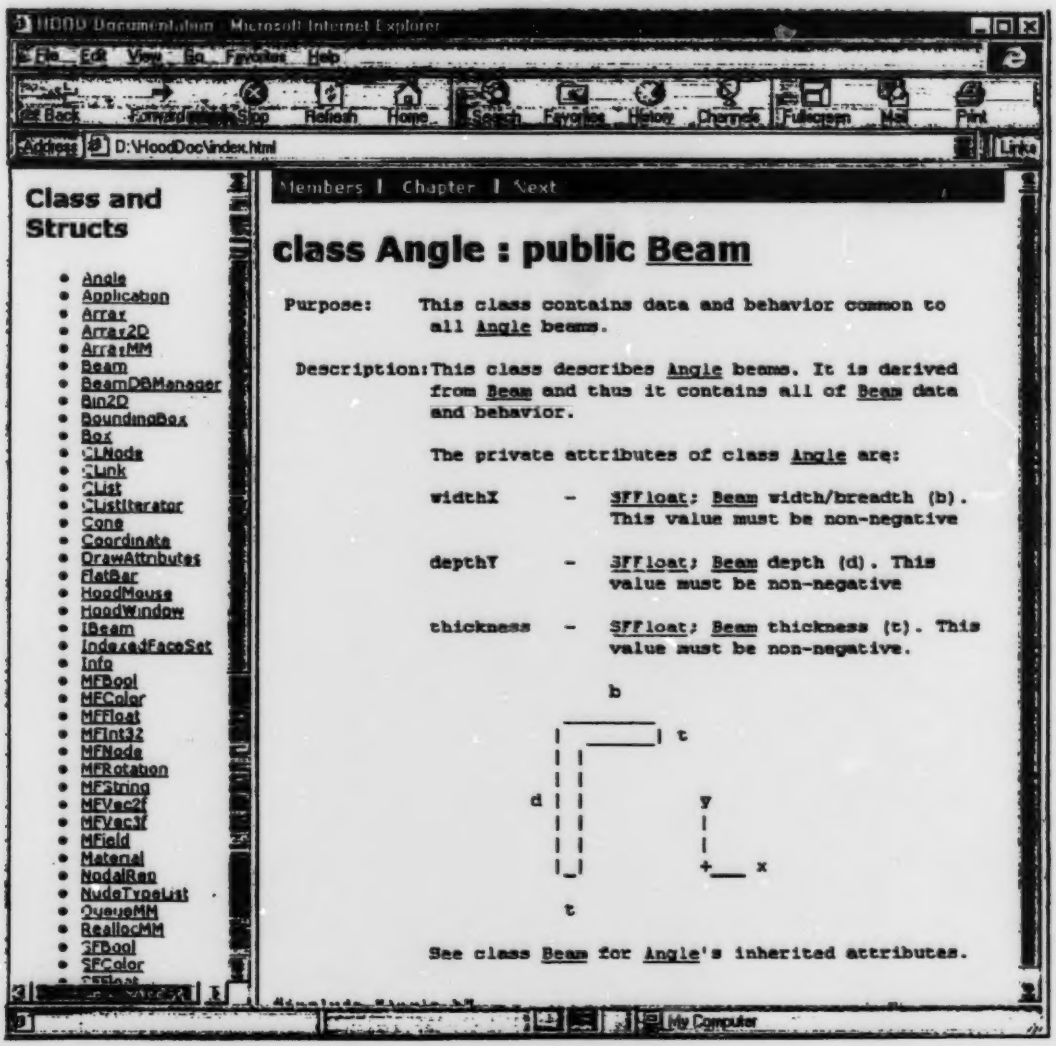

Figure 5: HOOD Object Documentation Example developed using Object Outline

5.3. Structural Finite Element Database

The finite element method (FEM) is to be used to undertake the structural analysis of the global ship structure and of structural details. A means to create the necessary finite element (FE) models from the geometric database is required. In the ISSMM concept development studies, a global MAESTRO [7] model of the Halifax Class was used in undertaking structural FE calculations. Local detail models have been produced with a specialized mesher (to be described later) and analyzed with the VAST [8] FE program. Any FE models that are produced for ISSMM or during the use of ISSMM should be stored in with reference to the main geometric database so that the models can easily be retrieved for future use or modification.

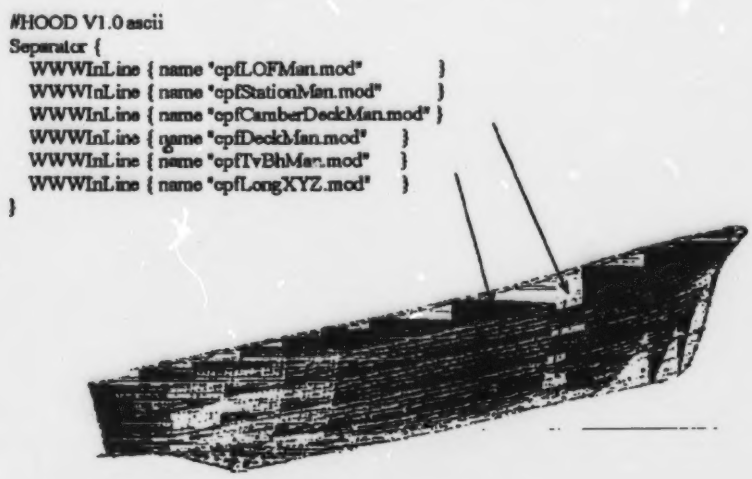
#HOOD V1.0 ascii
Separator {
WWWInLine { name "cpfLOFMan.mod" }
WWWInLine { name "cpfStationMan.mod" }
WWWInLine { name "cpfCamberDeckMan.mod" }
WWWInLine { name "cpfDeckMan.mod" }
WWWInLine { name "cpfTvBhMan.mod" }
WWWInLine { name "cpfLongXYZ.mod" }
}

Figure 6: Existing HOOD Data Manager for the Halifax Class

5.4. Material Database

A prerequisite for the successful application of an advanced numerical structural analysis method is access to relevant and comprehensive material property data. Under a previous ISSMM activity [9], through supplementary testing conducted by DREA, and through other related projects, a materials property database was created for the material systems employed during the Halifax Class fabrication. The material data should be stored in the overall HOOD geometric database and used through a material data manager.

While it has not been possible to quantify all of the materials properties for all of the Halifax Class materials systems, under all of the various possible conditions, the database contains entries for those properties that are of primary concern to future maintenance issues. The data was developed for the two primary structural base metal steels and several of the associated weld metals employed during the fabrication. The two base metals are CSA G40.21-350WT cat. 5 control rolled structural and ASTM A517 Grade F high tensile steel. The three primary weld metal systems evaluated were selected based

on the criticality of the resultant weld joints and the volume of welding done with these processes. These systems are;

a) (SMAW) 350WT / E7018 / 350WT
b) (SMAW) A517 / E11018M / A517
c) (FCAW) 350WT / E81-T1-Ni2 / 350WT

For each of the CPF materials systems of interest, the database includes entries on tensile strength data, true stress-strain curves, and work hardening characteristics. The testing was undertaken at two different temperatures and at static and intermediate strain rates. The testing conformed to the appropriate ASTM standard procedures. ASTM Standard E606 test procedures were employed to develop the cyclic stress - strain curves and the cyclic strain - life curves. The testing matrix was conducted at 2 different R (ε_{min} / ε_{max}) ratios (-1 and 0.5) and for two different environmental (air and seawater) conditions for each material.

The numerical prediction of the rate at which an existing defect or fatigue crack will propagate due to cyclic loading is accomplished through the application of established relationships between the change in crack length per load cycle (da/dN) and the crack-tip-stress-intensity factors (ΔK) during that load cycle. The simplest form of this relationship is referred to as the Paris-Erdogan equation. A number of similar relationships have been developed which account for stress ratio, mean stress, residential stress, threshold and other effects. The ASTM Standard E647 testing procedure was used to determine the fatigue crack growth coefficients (C and m) for the materials systems of interest. ASTM Standard E647 was also used to quantify the threshold ΔK values for the aforementioned R-values and environments.

The most important requirement for the safe operation of a naval platform is the prevention of cleavage fracture. In principle it is possible to use a fracture mechanics based approach with assumed, predicted or measured operating stress levels, and known material toughness parameters, to predict a maximum size of tolerable defect at a location of interest. Laboratory measurements of the relevant fracture mechanics parameters have been made using standardized ASTM procedures. There are numerous fracture parameters which describe the severity of a given defect under defined loading conditions (strain rate and temperature), including linear elastic K, elastic-plastic J, and crack tip opening displacement. Fracture toughness evaluations of the relevant parent metals, weld metals and heat affected zones have been conducted at three different temperatures (spanning upper and lower shelf behavior) and at two different loading rates (an intermediate strain and a dynamic strain rate). For each of the materials of interest, the relevant fracture toughness parameters have been derived from the experimental data.

5.5. Existing Tools

HOOD: [6] DREA has been developing the Hierarchical Object-Oriented Data (HOOD) manager system to use for ship structures. The HOOD system is under development and could be expanded to include the material property management for the ISSMM program. HOOD is written in C++ and is owned by the Crown.

MAESTRO: [7] MAESTRO is a global ship FE modeller and analysis program. It is marketed commercially by Proteus Engineering in Annapolis Maryland and DND has a license to operate the code through an informal exchange of technology agreement involving VAST and the MGDSA code. A new MAESTRO model of the Halifax Class was produced in 1998 and has been used as the main data structure for concept proof studies for ISSMM. While MAESTRO provides a good global FE analysis capability, it is not a suitable option for providing a global database for a ship. MAESTRO is written in Fortran and could be made available to developers of ISSMM by their own licensing arrangements with Proteus Engineering.

6. Operational Sea Loads

The sea loads algorithms in ISSMM will develop realistic load cases which are representative of CF ship operational scenarios (short and long term) for use in structural integrity assessment [1]. Ship operational profiles can be described in terms of the time spent in various operational cells that are a function of significant wave height, wave period, ship speed, heading to waves, and ship displacement. For each operational cell, the necessary load effects are determined, including hogging, sagging and lateral bending moments, shear forces, accelerations, hull pressures and still water bending moments. The operational cells contain data derived from linear response amplitude operators (RAOs) and, if possible, nonlinear sea keeping analyses. The operational profile load data are derived by a weighted combination of the various individual cells to give extreme and cumulative loads for ISSMM analysis of the defined operation. Sea trials [10] have been undertaken to validate the load prediction methods used in ISSMM concept development studies.

6.1. Structural Load Requirements

The basic load requirements of the structural analysis for ISSMM are extreme values and cumulative cycle histories of load effects throughout the hull structure for a given operational profile. Statistics in terms of coefficients of variation (COVs) and distribution types are also required to undertake probabilistic assessment of the residual strength. The structural analysis will be undertaken by finite element analysis, with the exception of the hull girder ultimate strength evaluation, which will use an integrated structural unit method (ISUM, described in Section 9) in conjunction with finite element analysis. Three categories of ship hull structural failure will be addressed by ISSMM. They each require specific load cases.

- **Hull girder strength:** extreme hull girder bending moments (BMs) at specific ship sections, including vertical hog and sag BMs (VBM), lateral BM (LBM), and the combined extreme VBM and LBM.

- **Component strength:** extreme applied forces on structural components during the operation, including the effects of BMs, shears, accelerations, and hull pressures.

- **Fatigue:** a representation of stress or strain amplitudes and their corresponding number of cycles for the structural detail of interest during the mission operation. Both crack initiation and crack propagation will be considered.

6.1.1 Still Water Bending Moment Load Requirements

The forces induced by the still water condition of the ship affect all of the structural failure modes. Still water bending moments (SWBM) resulting from the differences between ship weight and static buoyancy forces can be very significant, often as large or larger than wave induced forces. The Halifax class has several different loading conditions, varying from light ship to deep departure. Destroyers and frigates, unlike supply ships and commercial cargo vessels, do not have significant changes in weight during operations or even during life, with the exception of major refits where equipment may be changed. It is therefore proposed that only the deep departure still water condition be used for ISSMM. The weight distribution of the vessel and the displacement and trim will be stored in the Halifax class database. In cases such as midlife refits, where the still water bending moment may significantly change, a new set of load data will have to be generated reflecting the new condition of the vessel.

6.1.2 Hull Girder Ultimate Strength Load Requirements

The extreme applied bending moment may occur in the vertical plane (VBM), the lateral plane (LBM) or, more likely, in some plane comprising a combination of VBM and LBM. The extreme hog and sag VBM and extreme LBM values can be derived from operational cell data using extremal theory [1,11]. The extreme values of hog and sag VBM need to be augmented by the SWBM. The VBM may also include slamming and green sea loading effects, if nonlinear loads are calculated, as well as the shift in the mean bending moment due to nonlinearity (if the nonlinear sea load algorithms predict this).

The extreme combined VBM and LBM requires not only the extreme combined value, but also the relative angle between the two load components (position of neutral axis of bending). Determining this relative angle is not straightforward since there is no unique or average correlation between the vertical and lateral bending moments. Hence, a method has been proposed where the ratio of VBM to LBM is taken from the operational cell that produces the maximum combined BM [1].

6.1.3 Component Failure Load Requirements

A component may consist of a single plate panel, connection or a larger section of orthogonally stiffened grillage. The maximum force exerted on a component from the combination of hull pressure and acceleration load effects is required to evaluate component strength and stability. A modification of the American Bureau of Shipping Dynamic Load Approach (DLA) to include nonlinear loads and to be more specific for naval frigates is described in [1]. The basic premise of the DLA approach is that the extreme component loads for the ship mission occur in the same physical load situation (operational cell) as the maximum value of some dominant load effect. The dominant load effect would be a global quantity such as vertical bending moment, which can be easily determined through the sea keeping analysis. The assumption that the extreme component force will occur in the same physical situation, as does the maximum dominant load parameter, may be eliminated by using the spectral load method described in [12] for fatigue analysis. Using linear frequency domain loading in combination with spectral finite element analysis would allow identification of extreme component forces directly in the finite element analysis.

6.1.4 Fatigue Load Requirements

Fatigue analysis in ISSMM includes both crack initiation and crack propagation. Crack initiation requires the stress history at a specific point where the crack is expected to start. Crack growth requires the stress history over a region of structure where the crack will likely propagate in addition to the stress intensity at the crack tip.

Ideally the load information will provide a continuous life cycle history, but this is not practical as the ship operation is given in probabilistic terms and the sequence of future ship operations (headings, speeds, etc.) is not known. It is therefore necessary to develop some approximate method of representing the stress amplitudes and cycles. Stress amplitude sequencing, which is known to have an effect on fatigue crack initiation and growth, is not included in this approach.

Sea spectra data are derived from long term statistics and are therefore most applicable to predicting long term response. Using the sea spectra data to predict short term loads will require some care, particularly for fatigue, which is very sensitive to loading. Some additional short term load operations may be necessary for fatigue calculations.

There are several possible approaches to developing the stress cycle load information for fatigue analysis, these are discussed further in [1]. The following approaches have been considered for ISSMM and are briefly described below:

- Derive VBM exceedance curves from operational cells and derive stress exceedance curves as per the method of [13].
- Derive stress intensity transfer functions from VBM and LBM and combine these in each operational cell to get a histogram of stress, mean stress and lifetime cycles.
- Use a quasi-static spectral finite element structural analysis with pressure and acceleration frequency response functions as input to derive stress spectra.

The first two methods limit analysis to areas of structure which are affected mainly by primary hull girder bending load effects, while the last allows some scope for including all load effects. The latter method was investigated in an initial phase of ISSMM [14].

6.2. Operational Profiles

The operational profile will define time spent in each ocean area (Marsden Zone), and heading and speed probabilities, and hence the time spent in each separate operational cell (again, using the definition of cell as a unique combination of speed, heading, wave period and wave height). Two methods of entry for the operational profile are to be considered. A 'user defined' option will exist to define operational profiles for ISSMM. A 'default' set of long and short term operational profiles, based on current and expected CF ship deployment practice is also required. In addition, a set of profiles based on increasing severity of sea state should be generated in advance. These latter profiles would model the ship transiting a fixed sea state or a severe storm, thus allowing for the operator to determine the risk of operating in various sea conditions. A GUI for implementing the user-defined operational profile is being developed by Fleet Technology Ltd (FTL) in a current ISSMM contract. Figure 3 showed the current loads shell (GUI) prototype. Reference [15], a Ship Structure Committee (SSC) report, outlines an approach to developing operational profile statistics.

The BMT wave statistics package [16] has been purchased for use in ISSMM and is being included via an interface to the loads operational profile shell.

6.3. Response Amplitude Operators (RAOs)

Sea load analysis can be made more efficient by calculating the RAOs for the Halifax Class in advance and storing them in the ISSMM database. Two codes have been used in ISSMM concept studies for developing the necessary linear load data for the operational cells. The first is the 2-D strip theory code SHIPMO7 [17] which can supply bending moments, shears and accelerations. The second code is the three dimensional frequency domain potential flow code PRECAL [18] which can supply bending moments, shears, accelerations and hull pressure spectra. Comparisons of the results of the two codes have been made with trials data from CFAV QUEST [19] and HMCS NIPIGON [20]. Comparisons of ship motions, pressures and bending moments have been deemed to be acceptable for use in ISSMM. Figure 7 shows a comparison of pressures for HMCS NIPIGON and Figure 8 shows a comparison of vertical bending moments. PRECAL would be the preferred code, as it can supply pressures which are required for the DLA type of component strength analysis. PRECAL is proprietary to the Netherlands Ship Model Basin Cooperative Research Ships (NSMB CRS) organization to which DREA belongs. The RAOs will be developed as part of the database.

6.4. Nonlinear Sea Load Effects

Preferably, nonlinear time domain analysis would be used to generate the operational cell data in higher sea states. DND does not currently have a code which meets the necessary requirements. This technology is in a research stage worldwide. There are some possibilities that could be developed within the necessary timeframe to meet ISSMM requirements. Initially only linear codes will be used for ISSMM, which is the current standard for this type of analysis. The database should be structured in such a way that the cells could be populated with nonlinear results at a later date with little difficulty. The possibility of using model test RAOs for extreme seas should also be considered.

The CRS Sea Load Working Group (WG), in which DREA participates, is carrying out research on green seas and slamming loads. Results from this WG will be applicable to time domain codes. As with the nonlinear loads, the results used in the database should be able to be updated at any time to reflect improvements in the predictive capability.

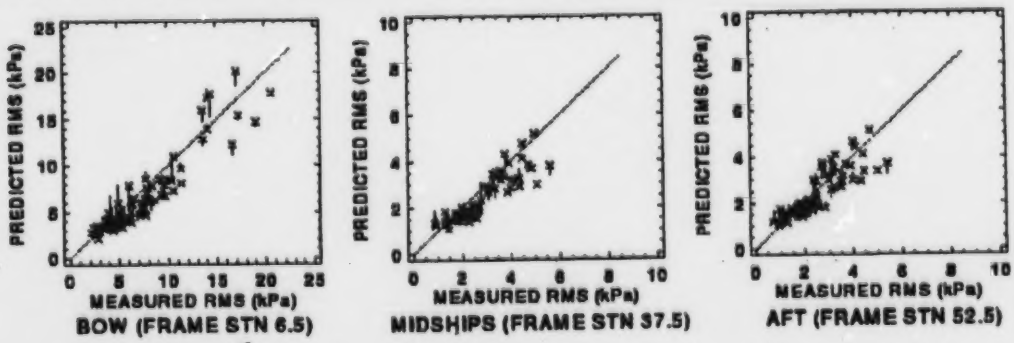

Figure 7: Comparison Of Measured And Predicted RMS Pressures For All Runs

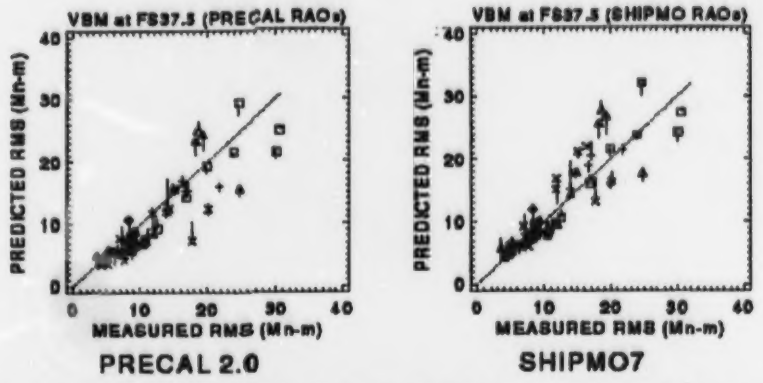
VBM at FS37.5 (PRECAL RAOs)
PREDICTED RMS (Mn-m)
MEASURED RMS (Mn-m)
PRECAL 2.0
VBM at FS37.5 (SHIPMO RAOs)
PREDICTED RMS (Mn-m)
MEASURED RMS (Mn-m)
SHIPMO7
Dominant Sea: □ Head, △ Bow, ◇ Beam, × Quart, + Follow

Figure 8: Comparison Of Vertical Bending Moment For All Runs At Frame 37.5

6.5. Load Library

Much of the sea load analysis for ISSMM can be done in advance to produce structural load cases for a set of operational profiles. A library of operational profiles and the necessary structural load case data for the global finite element models or ultimate strength models will be created and made accessible through the ISSMM GUI. A means to create new load cases for different operational profiles will also exist and these new cases will automatically be added to the ISSMM load library.

One necessary canned load case will be the Halifax Class **Design Load Case** as well as the current SSCP23 [21] 8-metre design wave.

6.6. Existing Tools

A number of existing tools have been described in the preceding text. Those and some additional ones are briefly summarized here:

Shipmo7: [17] This is a linear 2D sea keeping code produced and owned by the Crown and marketed nonexclusively by Fleet Technology Ltd. of Kanata. It will produce RAOs for ship motions and hull girder sectional loads. Validation through model tests and full scale ship trials have shown predictions to be acceptable for ISSMM structural calculations. The code is written in Fortran and is available from DREA for ISSMM development.

PRECAL: [18] This code is a 3D linear sea keeping code based on the panel method. It has been produced through the NSMB CRS organization who own proprietary rights to the code. DREA is a member of this organization and can use the code and have others use it on their behalf. PRECAL predicts motion, pressure, acceleration and sectional force RAOs. Validation has been done similar to SHIPMO7 with good results. PRECAL is written in Fortran and can be made available to contractors from DREA for the purpose of developing ISSMM.

ISSMM Loads Shell: This is software currently being developed for ISSMM by Fleet Technology Ltd. It incorporates an interactive map of the world which allows users to plot the ship's route or use two canned operational profiles. An example of the user GUI was shown in Figure 3. It allows the user to input operational shell data (heading, speed probabilities, etc) or use default values. There are interfaces to the BMT wave statistics [16] and a SHIPMO7 RAO database for the Halifax class as well as other RAO databases. The program undertakes extreme statistics calculations [11] and produces structural loads for finite element models. The software is to be produced in an object-oriented approach and is due by the end of 1999. The code is written in C++, is owned by the Crown and can be made available to ISSMM developers. It has some small components of commercial software (world map) which can be purchased inexpensively by the ISSMM developer.

MGDSA Loads Interface: [24] The MGDSA code, which is a user shell allowing top-down finite element analysis for the MAESTRO code, will be described in more detail in following sections. Currently, the UK Ministry of Defence (MOD) through the Defence Evaluation and Research Agency (DERA) Rosyth organization, is having Martec Ltd. of Halifax, develop a sea loads interface for MGDSA. This will incorporate the operation of SHIPMO7, PRECAL, THAFTS (a UK hydroelastic code) and LAMP (a USN nonlinear sea keeping code) directly from global FE models of ships. It will undertake extreme statistics and produce load cases for MAESTRO and other FE code full ship finite element models. This software is to be produced in 1999 and will be owned by the UK. The Canadian DND has access to this code through collaborative arrangements with the UK MOD. An example of the MGDSA user GUI was shown in Figure 3. Martec Ltd. and the Crown jointly own the MGDSA code. MGDSA and the loads interface are written in Fortran. The executable code for MGDSA and the interface could be obtained for the ISSMM developer, but it is undetermined at this time whether source code could be made available.

BMT Wave Statistics: [16] This package incorporates probabilities of wave height and direction for the entire world as a function of Marsden zones. The database and associated software is owned and is available from BMT. DREA currently holds 5 user licenses for the British Maritime Technology (BMT) wave statistics package. A licence could be transferred to the ISSMM developer for the sole purpose of developing the ISSMM software.

VSHIP: VSHIP is an animated visualization program written at DREA which will simulate sea state and ship response in three dimensions. It allows the user to change the sea state and ship operational conditions via on-screen controls. It currently uses SHIPMO7 RAOs and a time-history post-processor. The program runs effectively on a PC (200 MHz or faster). The Crown owns this code. A sample screen shot is shown in Figure 9. It is written in object-oriented C++ and utilizes objects from the HOOD toolkit. This code could be made available to the ISSMM developer for the purposes of developing ISSMM.

7. Global/Detail Structural Analysis

The structural analysis for ISSMM will provide the means to translate loads to load effects in the intact or damaged ship structure. These load effects are moments, stresses, strains, displacements, and forces, which will be passed on to the ultimate strength, component strength and stability, and fatigue evaluation modules. Structural and material data will be supplied from the Halifax Class database.

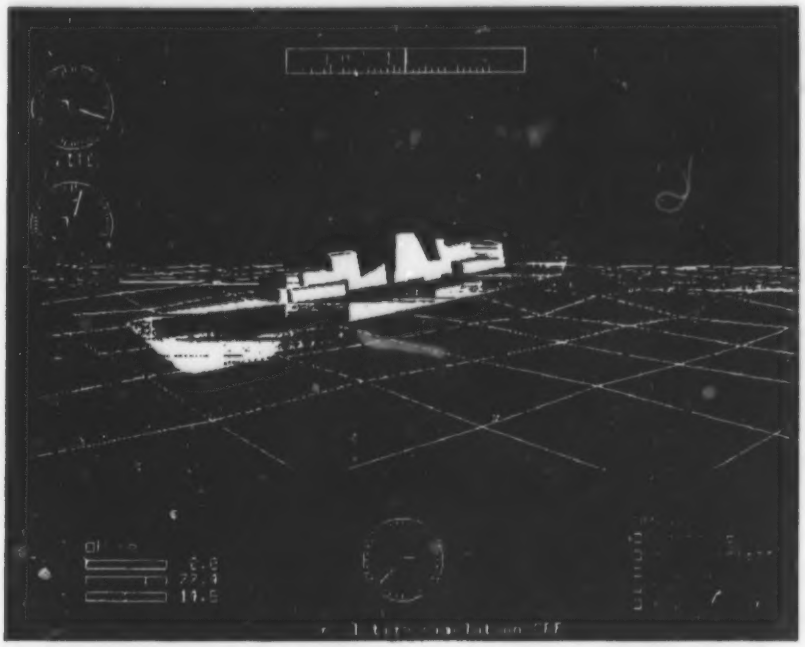

Figure 9: The VSHIP code for Visualizing Animated Sea State and Ship Response

7.1. Global Finite Element Model

The ISSMM software will produce a global finite element model of the Halifax Class from the information stored in the geometric ship database and data management system. The global model will not include finite element representation of structural details, but will include all shell plating, decks, longitudinals and transverse bulkheads, superstructure, frames, longitudinal stiffeners, tanks, floors, material data and weight distribution. Suitable unit load and vibration analyses should be developed and undertaken on the model to prove its validity and integrity. Model integrity tests such as element aspect ratio, disconnected lines, etc. should be provided and undertaken on the model. Graphical capabilities to view, edit and interrogate (for element type, properties, location and dimensions) the model and to view displacement and stress results should also be included.

In keeping with the automated and minimal user input requirements to run the ISSMM program, this model should be produced and validated as part of the ISSMM deliverable. A means to automatically update and track changes to the model through the

geometric data management system also needs to be created. It is likely that different versions of the model will be necessary to reflect different ships in the Halifax Class.

A MAESTRO finite element model of the Halifax Class has been previously developed (it is shown in Figure 3). This model can be made available to aid in producing the ISSMM global FE model from the database. The MAESTRO model is probably not adequate for the purposes of ISSMM as it lacks the hierarchical relational requirements and the detail information to produce both global and detail models.

7.2. Global Finite Element Analysis and Application of Loads

Linear and nonlinear FEA will be the main method of structural analysis used for ISSMM. The FE method is currently available in DND through the DND owned code VAST and the commercial code MAESTRO. The finite element models will have to be capable of accepting the following quasi-static load cases (including combinations of the following):

- sectional moments - both vertical and lateral,
- sectional shears,
- regular wave profile which will be specified by heading, height, length and crest position relative to the forward perpendicular (FP),
- pressure distribution over the ship hull, and
- rigid body accelerations of heave, pitch, surge, sway, roll, and yaw.

In addition, the finite element model will have to accept frequency domain responses for the hull surface pressures and rigid body accelerations. The program also needs to provide the capability to automatically apply correcting accelerations to achieve force balance.

The global finite element analysis will provide input displacement values for the top-down FEA of the structural details as described in section 7.4.

It is essential that the global analysis and load application be done automatically for the user. The user will only have to specify the desired operational scenario and the necessary global finite element analysis will be automatically undertaken.

7.3. Smart Structural Detail Model Generation

The objective of ISSMM is to determine the effects of damage on hull strength (ultimate hull girder and component strength) and endurance (fatigue). This requires finite element models which accurately model structural details such as cutouts and connections and the effects of damage on these details. Two main requirements of the detail models are that they be created in an efficient manner and that they accurately model the desired response. The user should not have to be an experienced FE modeller to create and analyze structural details.

The most efficient means of undertaking detail analysis is to have the required detail FE models already in a database. Fine mesh models of many of the critical structure details on the Halifax Class have been created as part of the initial ISSMM work. This has been done through a specialized detail-object meshing algorithm developed by DREA called DASS (Detail Analysis of Ship Structure)[22,23]. DASS was created to reduce the time to produce validated ship structural detail models to minutes instead of the days that are presently required using general purpose FE meshers.

Ships are usually built to a defined set of details which are documented by classification societies, owners (such as Navies), or the builders. Details include stringer and frame intersections, bulkhead and stiffener connections (watertight and non-watertight), penetrations, cutouts, etc. The defined detail may vary in component dimensions (plate thickness, web height, etc.) and possibly have minor variations such as the inclusion of tripping brackets. Details can thus be defined by a set of 2D patches which are defined by boundary lines (the DASS approach uses interior as well as exterior boundary lines to include cutouts and crack lines) and connected together at common boundaries to form the 3D detail. The 2D patches are meshed based on a description of the node distribution on the boundary lines consistent with a special purpose FE mesher.

Parametric stress analysis of the detail classes is undertaken to produce the best set of predefined nodal distributions for meshing the details. The nodal distributions may be defined by functions of the detail geometry (eg. a function of plate thickness or cutout dimension) as opposed to fixed values.

The detail class descriptions and meshing parameters are to be stored in the Halifax Class database and can be recalled during a ship analysis to quickly generate the detail FE mesh of interest. As the object-oriented database management system is developed, the specific detail classes will be identified at their locations in the global ship model and include the actual scantling information for that detail. The DASS approach to detail meshing is illustrated in Figure 10. Thirty structural detail classes have been developed for the Halifax Class [25]. Figure 11 shows two of these.

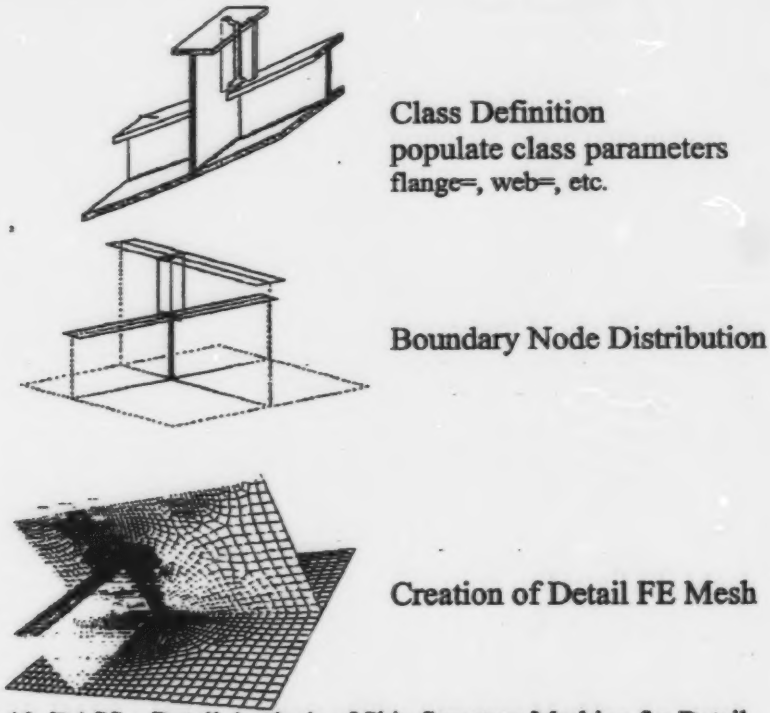

Figure 10: DASS – Detail Analysis of Ship Structure Meshing for Detail

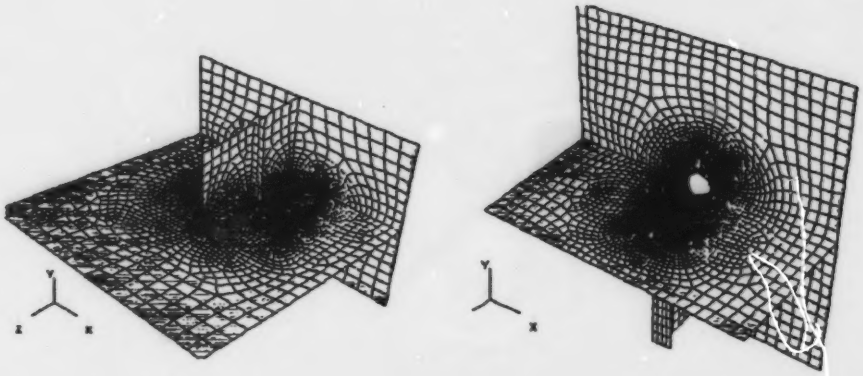

Figure 11: Some Detail Meshes Produced with DASS for the Halifax Class

In addition to the automated mesh generator for specific detail classes, a 'smart' mesher, based on the DASS system and HOOD toolkit, should be included in the ISSMM suite of programs. This algorithm would automatically remesh a given structure to improve results based on an initial stress analysis.

7.4. Top-Down FEA

The structural analysis will be based on a global coarse mesh model of the entire ship combined with fine mesh models of specific areas of interest. The areas of interest would be those which have sustained damage or where high stresses are shown to occur in the global analysis. The fine mesh model will be incorporated into the coarse mesh model by a top-down approach where displacement or force boundary conditions are derived from the global coarse mesh model and applied to the boundaries of a separate finite element model of the structural detail.

Figure 12 shows a detail model of the midship main deck and longitudinal bulkhead structure with part of a global MAESTRO model of the Halifax Class hull. The detail mesh is created with a set of boundary nodes which are common to the global model, and additional boundary nodes which are slaved to the global displacements through multi-point constraints.

Several developments have been undertaken with respect to developing and performing finite element analysis of structural details. The MGDSA [24] code provides an extensive array of options for undertaking top-down finite element analysis including model generation and result processing. This code has been developed by, and is jointly owned by, the Crown and Martec Ltd. MGDSA has been marketed to and is being used by several ship classification societies, other Navies, and a considerable number of other users. While MGDSA has provided extensive improvements in effectively undertaking FE analysis of ship structural details, it requires a finite element and structural analysis expert to properly use the code, and has many options to address a wide variety of ship types and structures, which are not required for ISSMM. As such, while MGDSA and ISSMM will have many common functions, ISSMM must be produced for less

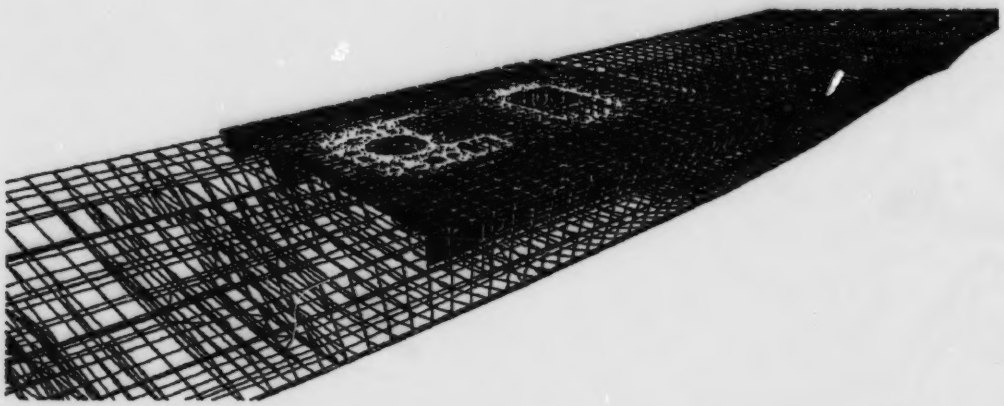

Figure 12: Detail Model of Halifax Class Midship Region in Top-Down Analysis

experienced users and provide more automation and efficiency in undertaking analysis specifically for the Halifax Class. In addition to being an international commercial success, MGDSA has been a successful development for testing the methods required for ISSMM. The ISSMM software will require the following functions to be developed and integrated for top-down finite element analysis:

a) Allow easy graphical and numerical identification of the local region of interest from the global geometric and finite element models.
b) Automatically extract all relevant information from the global model in creating the detail model.
c) Automatically create an intermediate finer mesh model (with default and/or user defined meshing parameters) of the global structure region of interest, including the option to model beams by shell elements.
d) Using the object-oriented DASS method (or acceptable alternative), automatically produce structural detail meshes for an existing 30-class library and develop and incorporate a GUI to produce and store additional detail classes in the Halifax Class detail library. The modeller will use a global object-oriented database manager to identify what types of details are in the region of interest. The automatic default meshes must be demonstrated to produce good stress results for fatigue crack initiation studies.
e) Automatically, or with minimal user input, provide continuity between the fine mesh detail model and the global model at the boundaries and undertake model integrity checks. Allow efficient semi-automated model editing facilities to correct deficiencies.
f) Automatically, or with minimal user input, transfer the global displacement boundary conditions and local loads from the global load analysis, to the boundary nodes of the detail model for top-down analysis, including automatic identification of the boundaries. This may incorporate 3 levels of FE mesh refinement; global full ship model, refined region of global model, and very refined region of the local area of interest.
g) Automatically undertake the local finite element analysis.

h) Provide facilities to graphically view displacement and stress results.
i) Extract and store the necessary results to undertake the structural integrity and risk assessments.

Considerable computational advantage could be gained by computing and storing the inverted stiffness matrix of the global model in advance as part of the database. The CPU intensive matrix formulation and inversion steps could be avoided and would allow analysis to proceed for various load cases for the global model and then on to the fine mesh models in a top-down analysis. It is also possible to create and store unit load transfer functions for the global model. Transfer functions between each single load component (pressure on a single panel, a rigid-body acceleration or a section bending moment) and all of the global finite element model displacements could be created and thus even further reduce computational time for the global model.

7.5. Model Checking and Result Display

While it is the intention of ISSMM to have as much automation and as little user input as practical, it must be recognized that undertaking finite element analysis of complex ship structure, particularly when incorporating damage, is a comprehensive and challenging task. It is impossible to envision all possible analyses ahead of time, and it is impossible to create automated procedures which will work perfectly in every modelling situation. To overcome this deficiency, it is necessary to develop an extensive set of both automatic and manual model integrity checks. These should be done both numerically and visually with 3D graphics. The automatic numerical checks will verify such things as element shape, element properties, model continuity, loads and application of boundary displacements. Deficiencies in the models should be graphically displayed to the user and a means provided to automatically correct the problems, much like a spell or grammar checker in a word processor.

The ISSMM software should allow efficient visualization, including mouse-drag model rotation, for the user to visually check elements, element properties, loads, boundary conditions and stress, strain, and displacement results.

7.6. Suggested Existing Tools

A number of existing tools have been described in the preceding text. Those and some additional ones are briefly summarized here:

VAST: [8] A Crown owned general-purpose finite element code, VAST has been used throughout the ISSMM concept development stage for linear, nonlinear and frequency domain analysis. It provides the detail FE capability in MGDSA. Martec Ltd. have done most of the development and nonexclusively market the code. VAST has comparable capabilities to other general purpose FE codes and can be provided at no cost to the ISSMM developer. It has been written in Fortran.

MAESTRO: [7] MAESTRO is a global ship FE modeller and analysis program. It is marketed commercially by Proteus Engineering in Annapolis Maryland and DND has a license to operate the code through an informal exchange of technology agreement involving VAST and the MGDSA code. A new MAESTRO model of the Halifax class

was produced in 1998. While MAESTRO provides a good global FE analysis capability, it is not a suitable option for providing a global database for a ship. MAESTRO is written in Fortran and would be available to the ISSMM developer through license with Proteus Engineering. The MAESTRO CPF model would be made available.

MGDSA: [24] MGDSA is an analysis shell which facilitates the top-down FE analysis of ship structure. It was originally developed for MAESTRO global models and VAST detail models but has expanded to include other FE codes. It incorporates a broad range of structural analysis capabilities including fatigue, ultimate strength, masts, underwater explosion response (UNDEX), pipelines, and others. It has been used as the test-bed for ISSMM concept development analyses. In general, MGDSA analysis and data management is at a level which is more user intensive than is required for ISSMM. MGDSA has intellectual property (IP) which is shared between Martec Ltd and the Crown. It is written in Fortran and an executable version could be made available for the purposes of the ISSMM development.

DASS: [22] The Detail Analysis of Ship Structure (DASS) approach to modelling and analyzing ship structural details based on 'smart' ship objects and details has been described above. This technology was developed and is owned by the Crown (notice of invention held). It is programmed in C++ and could be made available to the ISSMM developer for the purposes of the contract.

8. Damage Modelling and Analysis

The main objective of ISSMM is to assess the effect of damage on the structural integrity and operational capability of a CF vessel. The methods of determining the loads that a vessel will see during a specific operation and of assessing its structural integrity, described in the previous sections, are approximate in that they involve some uncertainty and use numerical methods which do not provide exact solutions. The assessment of the effects of damage will therefore be done as a comparison of the damaged case against the undamaged case for the given operation. This approach will reduce the uncertainties and approximations in the analysis to the greatest extent possible. Damage will be incorporated into the structural models and evaluated against the fatigue, ultimate strength and component failure modes. The damage to be considered in ISSMM will be:

Cracks: Propagation of detected cracks will be incorporated, as well as the effect of loss of structural continuity from a crack on ultimate strength and component failures.

Corrosion: The effect of loss of cross sectional area on component strength and stability and on the hull girder ultimate strength will be investigated. Corroded areas, which result in high stress concentrations, will be incorporated for possible crack initiation and subsequent propagation during the operation.

Deformation (Dents): The effect of changes in geometry caused by dents in the structure on component failure will be incorporated. The effect of imperfections produced by dents on component load-shortening curves will be considered in terms of hull girder ultimate

strength. Deformations, which result in high stress concentrations, will be investigated for possible crack initiation and subsequent propagation during the operation.

Loss of Structure: Damage cases (possibly by weapons or severe collisions) which result in loss of structure, will be modelled by removing components of the structural models. The effect of loss of structure on ultimate strength and on load distribution for component strength will be evaluated. Structural discontinuities that may result in high stress concentrations will be investigated for possible crack initiation and subsequent propagation during the operation.

It is important in developing ISSMM to envision a range of possible damage scenarios. A number of test cases will be developed to verify the capabilities. Some possible ones are outlined in Reference [1].

Damage will be modelled by altering the detailed finite element meshes of local structure to reflect the type of damage. This is not a straightforward process with somewhat conflicting constraints. One constraint will be that the damage must be incorporated as easily as possible, with minimal user input. This will require a predefined set of generic damage models which the user will select as the one most closely resembling the true case. Semi-automated finite element meshers will be developed to produce models which represent estimates of reported ship damage. The second constraint will be that the damage-model represents correctly the effect of the damage on the ship's structural integrity. Using generic damage models may not be suitable in some cases, and compromises will have to be made between accuracy and ease of use. The occurrence of damage can induce significant plasticity and residual stresses in the surrounding structure. It will not be possible to model this effect without modelling the damage process, a capability which is not within the scope of ISSMM.

8.1. Corrosion Modelling

Corrosion causes a reduction in both local and global structural section properties. Both the extent of the corrosion and the pattern will affect the structural response. The pattern of the corrosion could induce buckling modes of similar configuration to that of the corrosion.

Corrosion is measured during structural surveys at a number of discrete locations. A method to translate those measurements to an appropriate change in the structural finite element model will have to be created.

8.2. Deformation Modelling

Deformation of structure could occur locally through accidental denting of structure or on a more global scale resulting from severe global loads. It would be difficult to automate modelling of global deformations which change the shape of the global hull girder. This would probably require adjusting the global CPF database and would best be done as a special case if necessary.

Modelling of local structure deformation could be done in several ways. When the deformation has been well defined through measurements, an option to adjust the finite element nodal coordinates to match the measured shape should be created. Where deformation is only known in a general sense such as maximum deformation and the

extent of the damaged region, a generic modeller using algebraic functions should be created to change the detail finite element mesh. Deformation which results in tearing of the structure should be handled as a combination of altering the finite element mesh and removing and altering elements as discussed below.

8.3. Crack Modelling

Fatigue cracks will be modelled with finite elements. The crack tip will be modelled by two-dimensional crack tip elements while the immediately surrounding plate will be modelled by compatible stress elements. The crack and immediately surrounding structure will be created as a separate detail mesh which will be used in at top-down analysis. The DASS meshing algorithms have had initial developments towards incorporating crack lines in any detail model. In conjunction with the finite element code and a crack growth algorithm, the finite element model will be incrementally changed to reflect the new crack length and location of the crack tip. The process of analyzing the crack growth is further discussed in the section on fatigue. Figure 13 shows two examples of a crack meshed into a detail.

8.4. Assessing the Effects of Damage

The effects of the damaged structure on the ship's structural integrity will be evaluated through reassessment of the ultimate strength, fatigue, and component strength and stability limit states. This will be done primarily by comparison of the same analyses with and without the damage included in the finite element and ultimate strength models. A comparison of the structural capacities in the intact and damaged conditions and the change in probabilities of failure when considering the specified operation will be used to assess the effects of damage.

In the case of detected fatigue cracks, an assessment will be made on the probability of the crack becoming critical during the intended vessel operation.

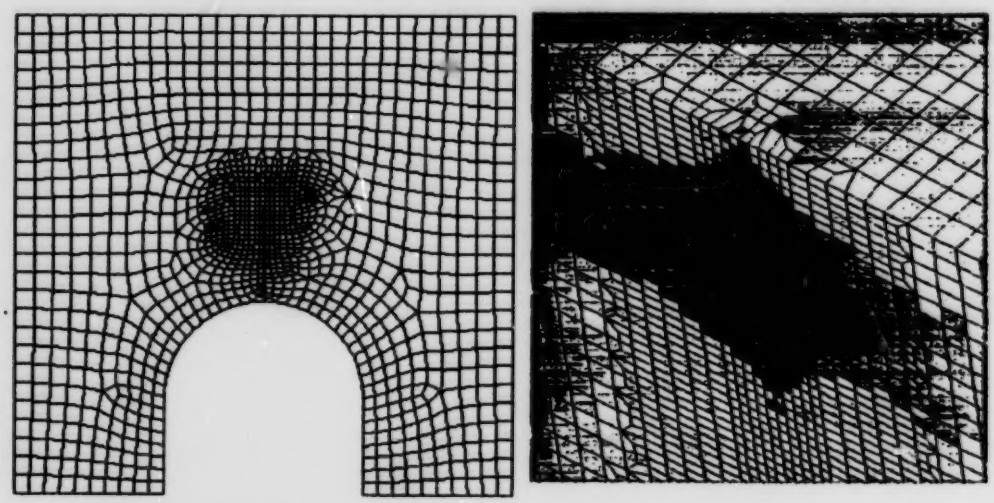

Figure 13: Examples of Crack Lines Meshed into Structural Details

8.5. Considerations for Modelling Gross (Weapons) Damage

Where damage has resulted in the complete loss of structure, local tearing of plates or significant fracture of plating or stiffeners, the finite elements representing that structure should be removed. An option to adjust and refine the mesh surrounding structure loss should also be included. In addition to the effect of loss of structure on section properties, the possibility of fatigue cracks initiating and propagating from areas of high stress concentration in the damaged area needs to be considered. Figure 14 shows an example of gross structural damage.

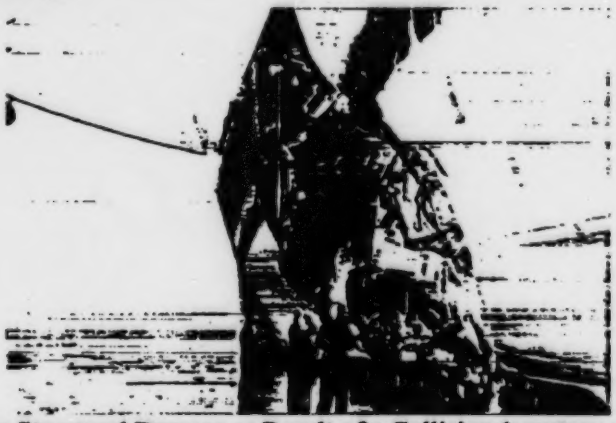

Figure 14: Gross Structural Damage as Result of a Collision (courtesy of U.K. MOD)

In cases where significant portions of structure have been lost such as the entire bow, it will be necessary to re-evaluate the hydrodynamic performance of the vessel. It will certainly be necessary to modify the global structural model to determine the redistribution of forces through the structure.

8.6. Existing Tools

There has been little development to date on developing specific modelling or analysis tools for damage cases. Proof of concept studies for corrosion have been undertaken [26] using the MGDSA code with general purpose model editors and generators. Figure 15 shows the full Halifax Class model with a simulated corroded region and a close up of the region with stress contours. Two very general crack meshing templates were created in MGDSA to develop and evaluate the spectral loads approach to crack analysis [14]. The inclusion of crack lines in the DASS modelling program has been investigated and appears to be a good approach to creating and propagating cracks.

One of the requirements for the meshing parameters for the 30 Halifax Class details [25] was that they produce FE meshes suitable for undertaking hot spot crack initiation analysis. These meshes will have to be adjusted as damage is incorporated.

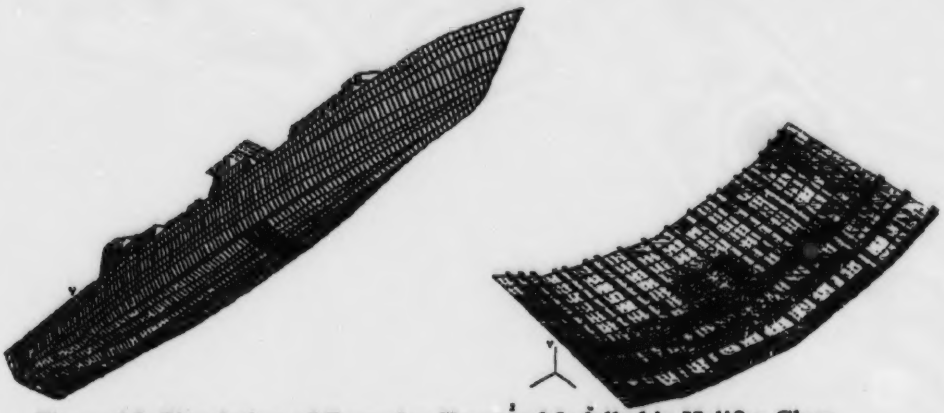

Figure 15: Simulation of Corrosion Damage Modelled in Halifax Class

9. Ultimate Strength Assessment

For ISSMM, ultimate strength will be defined as the maximum bending moment resistance of the ship hull girder structure, as opposed to component failure which is sometimes used to define ultimate strength. Failure of structural components will be undertaken in the component failure section of ISSMM. Although usually dominated by vertical bending, the ultimate strength limit should be calculated for combined vertical and lateral bending moments of the hull girder. Ultimate strength should also be calculated at several locations along the hull girder, particularly at stations affected by damage.

There are several options for undertaking ultimate strength analysis ranging from quick approximate methods to lengthy nonlinear finite element calculations with general-purpose finite element codes. Ultimate strength continues to be a topic of research as none of these methods have proven to be wholly satisfactory.

9.1. Scope

The ultimate strength analysis in ISSMM will determine the ultimate strength resistance of damaged and undamaged hull cross-sections for comparison to the expected extreme sectional bending moment value derived from the intended ship operation. The necessary input data to the ultimate strength calculations will be automatically derived from the Halifax Class global database and data management system including the nonlinear elasto-plastic material behaviour. The ISSMM code should allow the user to identify the desired region of the ship, automatically extract the cross-section information and automatically produce the structural unit divisions and data. The code will also extract all of the necessary load-shortening curve data from the database or produce new load-shortening data through nonlinear FEA, if required. The code will facilitate the insertion of damage to the cross-section and will launch the ultimate strength solver and present results.

9.2. Ultimate Strength Modelling Assumptions

Two programs, ULTMAT [27] and FABSTRAN/NS94 [28,29] have been used for ultimate strength calculations in the Department of National Defence (DND) [30]. The geometry of the cross section of interest and the elasto-plastic axial deformation characteristics of individual stiffened plate components which make up the cross section are required for these programs. The elasto-plastic characteristics of the individual stiffened plate components (load-shortening curves) can be generated from nonlinear finite element analysis or possibly from other numerical approaches and test data.

Both ULTMAT and FABSTRAN/NS94A use the idealized structural unit method (ISUM). The advantage of using this method is that its simplified two-dimensional modelling allows a fast calculation. The disadvantage is that its assumptions cannot fully represent the true behaviour of a three-dimensional ship structure. The idealized structural unit method models only the cross section, which is divided into structural units. Each unit contains a stiffener or a corner node (two or more plates meeting at an angle) and it's associated plating. Each unit has a length equal to the distance between transverse frames and is assumed to buckle between frames. It is assumed that there is no deformation of the transverse frames. Since each unit acts independently, it has its own load carrying characteristics. It can carry a certain amount of flexural buckling stress when subjected to compressive force, or yield stress when subjected to tensile force. Two major strength-reduction factors, initial imperfections and residual stresses, are considered through modification of the load-carrying characteristics of each individual structural unit. After the load-carrying curves (also called load-shortening curves) of each unit are known, the neutral axis of the cross section can be obtained by equilibrium of the axial force. The moment is then determined by summation of the force in each unit multiplied by its distance to the neutral axis. The major difference in calculations between the programs is how the load-carrying curves of each unit are derived. Figure 16 shows a typical ship cross section undergoing an ultimate strength collapse with the deck in compression.

9.3. Load-Shortening Curves

Regardless of which code is used for the hull girder ultimate strength evaluation, the ISUM methods require unit load-shortening curves. The load-shortening curve is an effective elasto-plastic, stress-strain collapse curve of a single-stiffened plate structural unit. For the intact Halifax class structure, load-shortening curves can be generated for several sections of the hull girder and stored in the ISSMM database. Many of these curves exist already for the Halifax Class through initial ISSMM work [31].

9.4. Integration of Damage Models

As mentioned in the previous section, the effect of damage on ultimate strength capacity can be evaluated by producing new load-shortening curves for the structural units affected by the damage. In extreme cases, or for conservative analysis, the structural units could be removed from the ultimate strength calculation. A means of modelling the damage (cracks, corrosion, dents) into the structural units will be required followed by a nonlinear FE calculation to produce the new load-shortening curve. A semi-automated 'smart' nonlinear FE solution algorithm will be required to meet the user requirements of ISSMM. This capability was investigated [32] with the nonlinear VAST FE code. It

would also be possible to produce some generic damage load-shortening curves in advance and store them as part of the ISSMM database.

Figure 16: Typical Ship Cross-Section Undergoing Ultimate Strength Collapse

9.5. Existing Tools

A number of existing tools have been described in the preceding text. They are briefly summarized here:

FABSTRAN/NS94: [28,29] The FABSTRAN/NS94 software system includes PREFAB, FABSTRAN, POSTFAB and NS94A, B, C and D. FABSTRAN (Frame And Beam Static and Transient Response ANalysis) is a nonlinear two-dimensional beam finite element program with built-in plate buckling algorithms which produce the effective stress-strain or load-shortening curves for the stiffener-plate units. By combining the load-shortening curves for each unit into the geometric information of the hull cross section, the NS94 programs calculate the full moment-curvature relationship for the cross section from which the ultimate moment capacity can be derived. NS94A is a version which undertakes vertical bending only and uses load-shortening curves from FABSTRAN. NS94B calculates bi-axial bending moment-curvature relationships and uses load-shortening curves from FABSTRAN. NS94C and NS94D undertake the same calculations as A and B, respectively, except that they use a set of generic load-shortening curve data rather than use FABSTRAN to calculate the specific curve for the cross-section. These codes were all produced and are owned by the UK MOD through Defence Evaluation and Research Agency (DERA) Rosyth. DND is currently contracting Martec Ltd to provide interfaces to these codes from the MGDSA code. Through international exchange agreement, DERA is providing the FABSTRAN/NS94 codes to DND for their use. The FABSTRAN/NS94 codes are written in Fortran. Permission for a contractor to use these codes in the development of ISSMM has been obtained from DERA Rosyth.

ULTMAT:[27] ULTMAT is a simplified version of the NS94 programs. It automatically divides the ship cross-section into units and assumes each unit buckles about its minor principal axis. The maximum load-carrying capacity of each unit is calculated with effective width concepts and column strength curves. Unlike, the NS94 codes which use load-shortening curves to produce complete moment-curvature calculations for the cross-section, ULTMAT only evaluates the maximum load carrying capacity of each unit, and gives only the collapse moment and not the full moment-curvature response. It can, however, calculate the plastic moment, and the maximum sagging and hogging collapse moment simultaneously. In addition, it provides the cross sectional constants, including warping and St. Venant torsion constants, and elastic stress information. A newer version of ULTMAT uses load-shortening curve data and is similar to the NS94D code. ULTMAT was developed by and is owned by the Crown (DREA). The initial version of the code was written in Fortran with a subsequent version produced in C++. The DREA scientist who generated this software is no longer employed at DREA and support is thus limited.

VAST: [8] A Crown owned general purpose finite element code, VAST has been used throughout the ISSMM concept development stage for linear, nonlinear and frequency domain analysis. VAST was augmented to provide 'smart' nonlinear analysis capability for generating the unit load-shortening curves. Martec Ltd have done most of the development and non-exclusively market the code. VAST has comparable capabilities to other general purpose FE codes and can be provided by DREA for the purpose of ISSMM at no cost. VAST is written in Fortran.

MGDSA: [24] MGDSA (Maestro Graphics/ Detail Stress Analysis) is an analysis shell which facilitates the top-down FE analysis of ship structure. It was originally developed for MAESTRO global models and results and VAST detail models and analysis but has expanded to include other FE codes. It incorporates a broad range of structural analysis capabilities including fatigue, ultimate strength, masts, UNDEX, pipelines, and others. It has been used as the test-bed for ISSMM concept development analyses. DND has contracted Martec to produce interfaces between MGDSA and the FABSTRAN/NS94 and ULTMAT codes to facilitate ultimate strength analysis. In general, MGDSA analysis and data management is at a level which is more user intensive than what is required for ISSMM. MGDSA has IP which is shared between Martec Ltd and the Crown. The code is written in Fortran. Executable code can be made available to the contractor for the purposes of ISSMM.

10. Fatigue Crack Assessment

ISSMM will require the development of, the integration of, and the validation of methods to assess naval platform structural fatigue crack initiation and growth. This will include both the probability of a crack initiating in an area of identified large stress concentration (possibly as a result of damage) and the probability of a detected crack propagating during a defined operation. Crack initiation requires the strain or stress history at the specific point where the crack is expected to start. Crack growth requires

the stress intensity at the crack tip and the strain or stress history over a region of structure where the crack will likely propagate.

Both crack initiation and crack growth are functions of stress level, number of cycles, and the stress history (or sequence). Ideally the load information would provide a continuous load cycle history, but this is not practical for ships as one does not know *a priori* the sequence of ship operations (headings, speed, etc.) and sea conditions. It is therefore necessary to develop some approximate method of representing the stress amplitudes and cycles using the probabilistic load data as described in [12,14]. Undertaking fatigue crack initiation and growth in response to stress or stress intensity spectra, which are produced from a defined ship operation, wave statistics and finite element analysis, is the most comprehensive approach to fatigue load definition available.

A material testing program has been undertaken [9] to develop the necessary material properties for crack initiation and crack growth models as well as those required for the strength models. Data has been developed for the A517 and 350WT base metals as well as for weld materials. The material property will be stored in the Halifax Class database. In addition, a report documenting the material data and its test procedure should be compiled as part of the ISSMM deliverable.

10.1. Scope

The ISSMM software will provide the appropriate finite element modelling and analysis to produce the necessary stress values and stress history, and the necessary analysis for fatigue crack initiation or crack propagation assessments for the area of interest (location of damage or high stress). ISSMM will also provide the necessary material information through the Halifax Class database and data management system. The software will allow the user to identify the detail area of interest, automatically produce the appropriate FE model, allow the user to efficiently modify the structure to incorporate damage (including cracks), and automatically undertake a fatigue crack initiation or growth assessment for the chosen ship operation. The challenge for ISSMM will be to make what is a complex analysis, easily available to the user in a 'black-box' format without excessive loss of accuracy to the methods.

10.2. Crack Initiation

It is desirable to allow the analyst the option of applying one or all of the three accepted approaches to fatigue crack initiation life prediction; nominal stress, hot spot stress and local notch tip stress/strain. The initial version of the ISSMM software should concentrate on the hot-spot method as it is most easily integrated with a FE based approach. The crack initiation life assessment will be undertaken for areas which are damaged or otherwise modified and may have resultant higher levels of stress, or for areas which are deemed to be susceptible to fatigue crack initiation from experience.

The software will allow two levels of analysis. The level 1 analysis will involve fatigue life predictions based on an extreme stress state at the various locations on the platform, without consideration of load spectrum or sequence effects. This level of analysis is intended to provide qualitative comparisons for the various platform connections to aid in refining inspection priorities. The level 2 analysis will involve fatigue life predictions based on the application of long term load spectrum information.

This analysis level is intended to provide quantitative predictions for primary structural connections.

Further discussion of DND efforts in crack initiation and methods that may be employed in ISSMM are described in references [34,35].

10.2.3 Nominal Stress Approach

The nominal stress approach to fatigue crack initiation is based on using values of global or primary stresses in the region of the detail of interest. These global stress histories are then compared to SN data which are developed specifically for the detail geometry of interest. It is outside the scope of ISSMM to develop individual SN curves specifically for each Halifax class detail, but there are catalogues of existing SN curve data based on joint classification (e.g. BS5400 [36]) which could be used. Since each major connection and detail in the Halifax class will be identified and stored in the Halifax class object database, it would be relatively easy to include a fatigue joint identification label and the associated BS5400 SN curve data with the detail data.

Given the location of the detail of interest, the ISSMM software will automatically identify the proper global stress history for the chosen operational profile, the detail or joint classification from the data base, the appropriate SN data, and then undertake a Miner's sum (or some other rule) fatigue life assessment.

10.2.3 Hot Spot Stress Approach

The hot spot stress-life approach to fatigue crack initiation is based on predicting (or measuring) stress near the crack initiation site and extrapolating these values to the actual initiation site (weld toe, plate edge). The stress values obtained this way are then compared to base metal or weld-metal SN curves. The stress analysis (FE modelling) includes the effects of structural details such as connections, cutouts, stiffeners, plate thickness changes, etc., but does not involve modelling of the welds. The level of modelling described above for the 30 Halifax detail classes (Figure 12) through a top-down analysis should give adequate stress information for the hot-spot approach and was developed with the hot spot approach in mind. Algorithms to extract and extrapolate stress values from these structural detail analyses will have to be developed and applied.

For the chosen location of interest, the ISSMM software will automatically identify the detail classification, automatically produce the FE mesh of the detail and the appropriate links into the global model, and allow efficient modifications of the detail model to incorporate damage if desired. ISSMM will then automatically undertake the top-down analysis for the given operational profile and derive the appropriate stress history or spectrum, automatically identify the necessary SN curve (or curves) and then automatically undertake the crack initiation analysis.

10.2.3 Local Notch-Tip Strain Approach

The local notch-tip strain approach is similar to the hot spot approach, but extends the level of detail to determining the stress or strain at the weld defect or notch. Analytical predictions of the stress state at a notch requires extensive three-dimensional solid modelling of the weld which is beyond the scope of ISSMM. In order to make this approach effectively available to the ISSMM user, a stress concentration or transfer function can be determined which will multiply the hot spot stress value (history or

spectrum) determined as for the hot spot approach. This stress concentration factor (SCF) could be determined from analytical functions and could be for a typical worse case scenario for the Halifax Class.

The ISSMM software would operate in the same way as for the hot spot approach, but would include the SCF and the cyclic stress-strain data for the material. This approach may be left as a later add-on to the ISSMM software.

10.3. Crack Growth

The ISSMM software will provide the capability to model the approximate growth of a detected crack in the Halifax Class structure. Due to the considerable uncertainties associated with fatigue crack calculations, the general philosophy of ISSMM will be to determine whether there is any reasonable likelihood that a crack will affect the structural integrity during the intended operation. Monitoring and reporting the actual physical crack growth progress during the ship operation will be an essential part of the ISSMM analysis.

Crack growth is modelled by determining the stress intensity at a crack tip, the associated number of stress cycles or stress spectra, and using this information in a particular crack growth law. The crack growth rate depends on the stress state of the material which the crack itself influences, therefore an iterative solution scheme is necessary. Finite element models can be used to predict the stress intensity value at the crack tip and the stress field around the crack. The load history can be determined using spectral FE analysis with the operational profile. These approaches are discussed further in [1,12].

Finite element modelling is the most suitable method to use to determine stress (or strain) and stress intensity values in ship structural details. A detail finite element model of the crack tip region, including elements which calculate the stress intensity at the crack tip from the local stress field, can be used. Loading is applied to the detail finite element model from the global FE model through a top-down analysis. The crack tip stress intensity and the loading frequency are then used in a particular crack growth law to determine how far the crack will propagate. As the crack propagates, the detail mesh at the crack tip and its immediate surroundings is remeshed to reflect the change in geometry. In order to minimize the uncertainties associated with meshing, a semi-automated smart meshing approach is required for detail and crack tip meshes which are appropriate to the fatigue models being used. As described above in Section 8.3 and shown in Figure 12, the DASS mesher can be expanded to include crack meshes.

10.4. Fatigue Calculations using Extreme Load

Load requirements for fatigue were discussed in Section 6. Ideally, a stress history or stress spectrum will be developed from the operational profile and the stress analysis. As a simplified approach to fatigue loading, an approximation can be made where stress data from a single static load analysis is used. SSCP23 [21] suggests an approach based on the 8-metre wave load, where an assumed spectrum is applied to the resultant stress. An approach similar to this should be developed for ISSMM. A fatigue initiation assessment based on a single extreme load will also be useful in determining critical structural areas and aid in developing inspection plans for the Halifax class.

A simplified, conservative alternative to undertaking a full crack growth analysis using a stress spectrum or time history, is to make an assessment of crack growth based on some likely extreme static load. In this case, the crack tip stress intensity value would be compared to the material critical stress intensity value.

10.5. Fatigue Calculations using Operational Load Spectrum

The load requirements to produce a stress history or stress spectrum for a given operational profile were discussed in Section 6 and references [1,12]. A means to order rationally or conservatively the occurrence of amplitude ranges in the stress history was given preliminary investigation in reference [14] and should be incorporated into the ISSMM software. The 'canned' operational profile load cases should have associated predefined load histories in the database. These can be used in conjunction with the stored global unit-pressure transfer functions to produce load histories for the detail area of concern.

10.6. Residential Strain Effects

In addition to the stresses and strains and the stress intensity values in the region of a hot-spot or crack tip caused by the sea loads and still water loading, there are also built-in strains caused by the manufacturing process of the structure. Welding produces residual strains and fitting together of structural components produces restraint strains. These two effects together with the still water strains are termed the residential strains. The residential strains can be added to the wave-induced strains in the form of a shift in the local mean strain.

The difficulty lies in determining the residential strains. It is planned that a residential strain atlas of the Halifax Class be developed using the Miniature X-Ray Diffraction (MXRD) system, developed by DREA and Proto Manufacturing [37]. The residential strain data will be stored in the Halifax Class database and used in the crack initiation and crack growth analyses.

The ISSMM software should provide the option to make appropriate corrections for mean stress and residential stress effects and environmental temperature effects if the data is available for the connection under analysis.

10.7. Existing Tools

The following software tools have been used in the initial investigation of fatigue crack initiation and growth for ISSMM.

VAST: [8] A Crown owned general-purpose finite element code, VAST has been used throughout the ISSMM concept development stage for linear, nonlinear and frequency domain analysis. VAST was augmented to allow frequency domain stress analysis given input pressure and inertia load spectra from PRECAL. A four-noded 2-D fracture element was also produced to provide a compatible crack-tip modelling capability. Martec Ltd. have done most of the development and nonexclusively market the code. VAST is written in Fortran and can be made available to the developers of ISSMM at no cost.

LIFE3D: [33] LIFE3D is a finite element based computer code for calculating crack initiation or growth rate. It was initially developed for through thickness crack

penetration but has been extended to the case of a through-thickness crack propagating across a plate and for crack initiation predictions. The program uses a detail finite element model of the crack tip region including special elements which calculate the 2-D stress intensity at the crack tip from the local stress field. The crack tip stress intensity and the loading frequency are then used in a particular crack growth law such as Paris's Law to determine how far the crack will propagate. As the crack propagates, the detail mesh at the crack tip and its immediate surroundings is remeshed to reflect the change in geometry. The code was developed by Martec Ltd. for the Crown. The Crown owns the code. LIFE3D is written in Fortran and can be made available to the developers of ISSMM at no cost.

MGDSA: [24] MGDSA is an analysis shell which facilitates the top-down FE analysis of ship structure. It was originally developed for MAESTRO global models and results and VAST detail models and analysis but has expanded to include other FE codes. It incorporates a broad range of structural analysis capabilities including fatigue, ultimate strength, masts, UNDEX, pipelines, and others. It has been used as the test-bed for ISSMM concept development analyses. In general, MGDSA analysis and data management is at a level which is more user intensive than what is required for ISSMM. MGDSA has IP which is shared between Martec Ltd. and the Crown. MGDSA is written in Fortran and the executable code can be made available at no cost for the purpose of developing ISSMM.

DASS: [22] The Detail Analysis of Ship Structure approach to modelling and analyzing ship structural details based on 'smart' ship objects and details has been described above in Section 7.3. It embodies the smart Halifax class detail generation which provides automated meshing of 30 typical Halifax class details to a level suitable for hot-spot stress analysis. Preliminary work on including cracks into detail meshes has also been included. This technology was developed and is owned by the Crown (notice of invention held). DASS is written in C++ and can be made freely available to the developers of ISSMM.

PRECAL: [18] This code is a 3D linear seakeeping code based on the panel method. It has been produced through the Netherlands Ship Model Basin (NSMB) Cooperative Research Ships (CRS) organization who own proprietary rights to the code. DREA is a member of this organization and can use the code and have others use it on their behalf. PRECAL predicts motion, pressure, acceleration and sectional force RAOs. PRECAL is written in Fortran and the executable code can be made available to the developers of ISSMM.

11. Structural Component Stress and Stability Assessment

The previous sections have dealt with fatigue and hull-girder-ultimate-strength failure modes. It is also necessary for ISSMM to address the performance of individual ship structural components. Component evaluation follows traditional ship design practice where the ship is evaluated as individual components rather than a continuous structure. Finite element analysis of ship structures lends itself to the evaluation of continuous structure rather than individual components. Using FEA to evaluate structural

components is a challenge in that the finite elements and resultant stresses must be mapped to the particular component that they represent. The MAESTRO program [7] successfully achieved this for global structure and gained considerable approval in the naval architecture community as a result.

The effects of damage on these components will be to either change the configuration of the component itself if it is directly damaged, or to change the load which is applied to the component as a result of redistribution of the load due to damage in adjacent structure. Strength and stability are the structural parameters which must be evaluated for each component.

11.1. Scope

The ISSMM software will allow the user to automatically identify ship structural components in a region of interest and automatically produce the FE mesh and map it to the component. The ISSMM user will then be able to modify the models of the components to include damage if desired, determine those component's stress and stability limits, and determine the loads (forces) that the component will see from the given operational profile. These load and resistance factors will then be provided to the strength and stability limit states.

11.2. Definition of Structural Components

The ship structural components that need to be evaluated are:

- longitudinal beams between frames, bulkheads, or major transverse structure,
- transverse beams between longitudinals,
- transverse beams between major longitudinal structure,
- single plate panels between longitudinals and transverse frames,
- single stiffened plate panels between frames,
- grillage systems consisting of orthogonally stiffened plating bordered by major transverse and longitudinal structure,
- transverse bulkheads,
- longitudinal bulkheads, and
- the above structure including cutouts and discontinuities.

This list is considerable but may not be exhaustive.

11.3. Stress and Buckling Stability Limits

The stress components determined by the FEA and the stress limits used for the particular material should be consistent with those used in DND naval design standards. Similarly, the limiting stresses or forces determined for the stability of the structural components should be derived using formulae consistent with those in DND design standards. Limiting values may also be determined using nonlinear elasto-plastic FEA collapse analysis of the components. Currently the DND design standard limits should be derived from the UK NES 154 and SSCP 23 documents [20,21].

In determining the applied force on structural components resulting from a specific operational profile, the top-down FEA approach should be used with the most likely extreme static global sea load determined for the operation.

11.4. Existing Tools

This area of ISSMM has not seen much activity during the concept development stage. Codes which do address component evaluation to some extent are:

MAESTRO: [7] MAESTRO is a global ship FE modeller and analysis program. It is marketed commercially by Proteus Engineering in Annapolis Maryland and DND has a license to operate the code through an informal exchange of technology agreement involving VAST and the MGDSA code. A new MAESTRO model of the Halifax class was produced in 1998 with the conclusion that while MAESTRO provides a good global FE analysis capability, it is not a suitable option for providing a global database for a ship. In terms of component evaluation, MAESTRO input data is in terms of ship strakes which consist of frames, girders and stiffened plate. MAESTRO automatically assigns and maps finite element meshes to these strakes. As a post processor to the FEA in MAESTRO, an evaluation module compares the FEA resultant stresses and forces in the strakes to design code limiting values for a range of strength and stability limits. MAESTRO is written in Fortran and can be licensed from Proteus Engineering.

Ultimate Strength Codes: The load-shortening curves developed for single stiffened plate structural units for use in the ultimate strength codes can be used to define ultimate collapse loads for these structural components. The integration of the ultimate strength codes into MGDSA includes a module to compare the collapse load limit derived in this way to the force applied to the component from an extreme static wave load. This capability is owned by the UK MOD and DND and can be made available for the development of ISSMM.

12. Mission Risk Assessment

The user of ISSMM will enter the desired ship operation and the structural condition of the ship, either defining damage using one of the damage modellers or specifying the intact condition. The likely loads that the ship will see during that operation, the structural capacity of the intact ship and the effects of damage have to be evaluated and presented to the user in a meaningful manner.

12.1. Probabilistic and Deterministic Risk Assessment Methods

Probabilistic or reliability methods of structural analysis are becoming accepted in structural engineering practice and are particularly useful in comparing the performance of two structural systems against a given load, particularly when that load is best described in statistical terms. A comparison of probabilities of failure between the damaged and undamaged cases will provide a tangible sense of the change in risk to the user.

A deterministic evaluation can be carried out using characteristic values of the load effects and resistance (such as the stress occurring in a panel from the finite element analysis versus the nominal yield stress of the material). The formulation of the working stress, safety factor approach to the various failure modes can be taken from the methods described in Section 11 for component failure.

12.2. Failure Limit States

It is recommended that component reliability analysis be used for ISSMM. This involves evaluating failure modes through limit state equations involving failure of individual structural components, rather than complex system failure involving multiple member failure. Component limit states for ship structures such as panel buckling, local yielding, stiffener failure, hull girder ultimate moment, and fatigue crack initiation and growth have been developed and are available from several sources such as [38,39]. The main difficulty for ISSMM is in providing the statistical input to the reliability calculations which will have to come from the load and strength modules. It is beyond the scope of ISSMM to develop load and strength statistics other than by reviewing what has already been developed and determining acceptable values.

A particular failing of reliability methods for ship structures to date is that calculated probabilities of failure have not been accepted in absolute terms. Uncertainties in loads and strength analysis and acceptable design limits of failure probabilities have not been satisfactorily determined to allow design by reliability methods. Reliability methods have proven to be a very powerful tool in comparing structural designs and in determining the relative effects of various design parameters. This fits the ISSMM scenario very well, where the relative safety of a vessel in a damaged condition is compared to the undamaged condition. It also provides a means to isolate the effects of specific parameters such as a corroded plate or vessel speed. Some of the input requirements for probabilistic assessment are statistical distributions and uncertainties for:

- extreme load effects from loads and structural analysis modules,
- lifetime load effects (stress and strain) from loads and structural analysis modules,
- yield strength and material properties,
- ultimate strength capacity and associated variables,
- buckling capacity and associated variables for component strength,
- fatigue resistance and associated variables, and
- displacement

Reliability algorithms are available in programs such as CALREL [40], etc. The approximate First Order Reliability Method (FORM), Advanced First Order Reliability Method (AFORM), and Second Order Reliability Method (SORM) routines exist in these codes as well as some higher order methods.

12.3. Presentation of Risk to the Operator

The risk assessment software will undertake deterministic and probabilistic assessment of the absolute and relative risk of operation for the damaged vessel. Information that can be used to assess risk has to be presented to the ISSMM user in a convenient and meaningful manner. Some of the possibilities for presentation are:

- Colour contour plots on the ship model which show where and by how much structural limits have been exceeded.
- Colour contour plots of stresses and displacements.
- Histograms of failure probabilities for specific limit states using different colours for the intact and damaged vessel.

- Histograms of safety factors for specific failure modes using different colours for the intact and damaged vessel.
- Histograms or curves showing the change in failure probabilities and safety factors with vessel speed, heading and significant wave height.
- Curves showing the progression of crack growth with time of operation.
- Screen printout file describing the various analyses and results produced by the ISSMM suite.

It is necessary to present the results as a function of ship operation in a manner to allow the user to determine what operational conditions should be avoided. A means to account for the repair of the damage to different levels in the analysis models and determine the effects of these repairs on the risk should also be developed and implemented.

13. Testing

The aim of ISSMM is to develop a tool to be used for operational decisions for the Halifax Class. As such, a considerable effort needs to be placed on ensuring that the software is properly produced and runs efficiently and that the functional methods used in the analysis produce usable results.

13.1. Software Integrity Testing

The developers of the software will have to demonstrate good quality control , software development practice, and documentation procedures for the software. Software must be demonstrated to be as efficient as possible for both wall clock and CPU time. An independent contractor should be employed to review the code development.

13.2. Software Functionality Testing

To ensure that each of the analysis functions runs correctly and that the various components properly operate with each other, a set of test problems should be developed to continually test and verify the operations of all the software modules as they are developed throughout the course of the project. Contractors developing software algorithms will be asked to verify and demonstrate their codes against specific problems. As above, an independent contractor should be employed to undertake this testing in addition to the development contractor (or contractors).

14. Documentation

14.1. User's Manual

The user's manual should contain a description of all program functions and step by step descriptions on how they are used. A number of example problems should be presented for common uses of the code.

14.2. Programmer's Software Management Manual

Documentation of all of the programming objects and how they can be used should be developed. The documentation should allow any competent programmer to modify and expand the code in the future.

14.3. On-line Help and Documentation

An on-line help function attached to all functional activities (buttons) in the code should be developed using standard web browsers. An explanation of what the feature is and how to use it should be given. In addition, short on-line tutorials on how to accomplish certain techniques should also be included. The on-line documentation would probably incorporate the user's manual.

14.4. Theory and Functionality Manual

A document describing the theory used in the functional modules (fatigue, ultimate strength, etc) is required.

14.5. Tutorial

A tutorial to train ISSMM users should be developed and provided by the ISSMM contractor and given at the completion of the project.

15. Existing ISSMM Contract Work

The following is a brief summary of contracts which have been undertaken to date for ISSMM, draft reports have been delivered for some of these and others are still underway:

'Finite Element Based Spectral Analysis Methods for Fatigue Crack Growth and Initiation of a Ship Operating in a Seaway', Martec Ltd., Halifax, Nova Scotia. – draft report delivered

'Materials Property Database for Canadian Patrol Frigate Steels and Weldments', Fleet Technology Ltd, Kanata, Ontario. – not yet complete

'Development of Specifications and Algorithms for Finite Element Meshes of Ship Structural Details', Fleet Technology Ltd, Kanata, Ontario. – draft report delivered

'Development of an Object Oriented Ship Database and Finite Element Modeller for Ship Structures', Martec Ltd., Halifax, Nova Scotia. – not yet complete

'Incorporation of UK NS94 Ultimate Strength Analysis in MGDSA', Martec Ltd., Halifax, Nova Scotia. – not to be completed until 2000

'Loads Software Shell', Fleet Technology Ltd, Kanata, Ontario. – not to be completed until 2000

16. Concluding Remarks

This technical memorandum has summarized the work undertaken to date towards producing the integrated software tool for the ISSMM project. This summary is not exhaustive in discussing the various technology areas but describes the methods and work undertaken in DND. The potential bidders for the ISSMM software development may choose to use the described approaches and tools or may decide to propose alternatives which provide similar functionality.

Overall there has been considerable progress made in the technologies which affect the ISSMM project. Some of these have been internationally recognized and are being used by several commercial and other naval ship authorities. In many respects Canadian industry and DREA have had the opportunity through the ISSMM project, to successfully lead world efforts in advancing the structural safety assessment of ships.

17. Acknowledgements

The work described in this memorandum is the result of efforts of many people: LCdr Tony Graham of DMSS 2 who is the ISSMM project manager; Orest Bluy and preceding him Richard Morchat, the ISSMM project directors; DREA scientists John Porter, David Stredulinsky, Layton Gilroy, Kevin McTaggart, Ross Graham and Sam Ando; and the several contractors who have contributed to ISSMM concept developments.

18. Reference Documents

1) Pegg N.G., `General Development Plan for the CRAD Major Development Project, Improved Ship Structures Maintenance Management - ISSMM' DREA TM 97/298, 1997.

2) Pegg N.G., Stredulinsky D.C., Gilroy L.E., Hu S.Z.,`DREA Pilot Study for Residual Strength of a Damaged Ship Structure' DREA TM 97/206, 1997.

3) Orisamolu I.R., Liu Q., Kumar R., Chernuka M.W., Norwood M.E., Bea R.G., `Pilot Study for Structural Risk Assessment Program', Martec Ltd, Halifax, DREA CR/93/470, 1993.

4) Srinivasan J., Kirkhope K., Bera K, Basu R., `Pilot Studies for Structural Risk Assessment Program (STRAP) – Final Report', Mil Systems Engineering Inc., DREA CR/93/475, 1993.

5) Pegg, N.G., Gibson, S., 'Application of Advanced Analysis Methods to the Life Cycle Management of Ship Structures', Advances in Marine Structures III, DERA, Scotland, 1997.

6) HOOD User's Manual, Martec Ltd Technical Report, TR 99 05, February, 1999.

7) MAESTRO - Method for Analysis Evaluation and Structural Optimization, User's Manual - Version 7.1.1', distributed by Proteus Engineering, Stevensville, Maryland, USA, January 1996.

8) VAST - Vibration and Strength Finite Element Code, User's Manual - Version 7.0, Martec Ltd., 1996.

9) Tensile Properties Of Cpf Hull Structural Materials, Fleet Technology Ltd Report, June 1997.

10) Pegg N.G., Stredulinsky D.C., Gilroy L.E.,'Summary of DREA Structural Load Trials on HMCS NIPIGON', DREA TM in review

11) McTaggart, K.A., 'Prediction of Extreme and Fatigue Sea Loads Using Linear Theory', DREA TM/98/215.

12) Stredulinsky, D.C. , 'Proposed Method for Calculation of Spectral Response to Random Wave Loading Using Top-Down Finite Element Modelling', DREA TM97/209, 1997.

13) Sikora J.P., Dinsenbacher A., Beach J.E., `A Method for Estimating Lifetime Loads and Fatigue Lives for SWATH and Conventional Monohull Ships', Naval Engineers Journal, May 1983.

14) 'Finite Element Based Spectral Analysis Methods for Fatigue Crack Growth and Initiation of a Ship Operating in a Seaway', Martec Limited, September, 1997.

15) Glen I.F., Paterson R.B., Luznik L., Sea Operational Profiles for Structural Reliability Assessments, Fleet Technology Ltd., DREA Contractor Report in review, 1998.

16) 'World Wide Wave Statistics', British Marine Technology.

17) Shipmo McTaggart K.A., ``SHIPMO7: An Updated Strip Theory Computer Program for Predicting Ship Motions and Sea Loads in Waves," DREA Technical Memorandum 96/243, 1996.

18) NSMB CRS PRECAL Working Group, 'User's Manual, Motion, Load, and Pressure Program Suite - PRECAL', BMT SeaTech CRS Report, January, 1995.

19) Stredulinsky, Pegg, Gilroy, 'Comparison of Motion and Load Predictions with Full Scale Trial Measurements on CFAV QUEST', Advances in Marine Structures III, DERA Rosyth, May 1997.

20) NES 154, Naval Engineering Standard, 'Design Standard for Surface Ship Structures, UK MOD, December 1989.

21) `Design of Surface Ship Structures', (SSCP 23), Volume 1 and 2, U.K. Ministry of Defence, 1989.

22) Pegg N.G., Heath D., Norwood M.E., 'Improved Ship Detail Finite Element Stress Analysis', Proceedings of PRADS 98, The Hague, September, 1998.

23) Kumar R., Pegg N.G., Heath D., Stone K., 'A Computerized Method for the Rapid Global and Local Analysis of Ship Structure', RINA Warships 98 Proceedings, London, June, 1998.

24) 'MAESTRO/DSA', Martec Ltd., DREA CR in review, October, 1998.

25) 'Development of Specifications and Algorithms for Finite Element Meshes of Ship Structural Details', Fleet Technologies Ltd., DREA CR in review, December, 1998.

26) Smith, D.R., 'Application of the MGDSA Program to Studying the Effects of Corrosion on a CF Warship', DREA Contractor Report in review.

27) Hu, S.Z., `EBMS - A Program for Equivalent Beam Modelling of Ship Structure', DREA Technical Memorandum 95/208, April 1995.

28) Dow, R.S. and Smith, C.S., `FABSTRAN : A Computer Program for Frame and Beam Static and Transient Response Analysis', Admiralty Research Establishment, ARE TR 86205, December 1986.

29) Dow, R.S., 'NS94C/D Ultimate Strength Design Method for Ships,' DRA Draft report.

30) Hu, Thomas S.Z., `Ultimate Strength of Halifax/Iroquois Class - Phase I', DREA Technical Memorandum 95/224, 1995.

31) Jiang L., Chernuka M.W., 'Continued Development of A Normalized Effective Stress-Strain Curve Database of Stiffened Plates for the Halifax Class Frigate, Martec Ltd., Report TR-98-08, March, 1998.

32) IVAST-An Efficient VAST Finite Element Program for Linear and Nonlinear Static Analysis', Martec Ltd, DREA CR in review, November, 1998.

33) Wallace, J.C., and Chernuka, M.W., `LIFE3D Three-dimensional Crack-growth Simulation Program Suite User's Manual', Martec Limited, July 1990.

34) Porter J.F., "Naval Platform Fatigue Considerations", DREA RN/DL/93/9, December 1993.

35) Porter J.F., Yee R., "Fatigue Crack Initiation Prediction for Naval Platform Weldments", 2nd CF/CRAD Meeting on Naval Applications of Materials Technology, pg 577-607, Halifax, N.S., May 1995.

36) BS5400, Part 10, Code of Practice for Fatigue, British Standards Institute, 1980.

37) Malik L., Yee R., "Applications of X-Ray Diffraction Stress Measurements in Ships", Fleet Technology Limited Contractor Report 3762C-11 submitted to DSE 5-5, NDHQ, Oct. 1993.

38) Mansour, A., Thayamballi, A., `Probability-Based Ship Design Loads and Load Combinations,' Ship Structure Committee Report SSC-373, 1994.

39) Mansour A., Lin M., Hovem L. Thayamballi A., `Probability-Based Ship Design Procedures: A Demonstration,' Ship Structure Committee Report SSC 368, September, 1992.

40) Liu P.L., Lin H.Z., Der Kiureghian A., `CALREL User Manual,' UCB/SEMM-89/18, 1989.

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM
(highest classification of Title, Abstract, Keywords)

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1. ORIGINATOR (the name and address of the organization preparing the document.. Organizations for whom the document was prepared, e.g. Establishment sponsoring a contractor's report, or tasking agency, are entered in section 8.)

DREA

2. SECURITY CLASSIFICATION (overall security classification of the document including special warning terms if applicable).

UNCLASSIFIED

3. TITLE (the complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S,C,R or U) in parentheses after the title).

Development of the ISSMM Software Tool for Residual Strength Assessment of Ship Structures

4. AUTHORS (Last name, first name, middle initial. If military, show rank, e.g. Doe, Maj. John E.)

Neil Pegg

5. DATE OF PUBLICATION (month and year of publication of document)	6a. NO. OF PAGES (total containing information Include Annexes, Appendices, etc).	6b. NO. OF REFS (total cited in document)
March 1999	40 (approx.)	40

7. DESCRIPTIVE NOTES (the category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered).

DREA TM

8. SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include address).

Defence Research Establishment Atlantic
PO Box 1012
Dartmouth, NS, Canada B2Y 3Z7

9a. PROJECT OR GRANT NO. (if appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant).	9b. CONTRACT NO. (if appropriate, the applicable number under which the document was written).
Project 1.g.d.11	

10a ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity. This number must be unique to this document.)	10b OTHER DOCUMENT NOs. (Any other numbers which may be assigned this document either by the originator or by the sponsor.)
	DREA TM- 1999-055

11. DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)

(x) Unlimited distribution
( ) Defence departments and defence contractors; further distribution only as approved
( ) Defence departments and Canadian defence contractors; further distribution only as approved
( ) Government departments and agencies; further distribution only as approved
( ) Defence departments; further distribution only as approved
( ) Other (please specify):

12. DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in (11) is possible, a wider announcement audience may be selected).

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM
UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM
(highest classification of Title, Abstract, Keywords)

13. ABSTRACT (a brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual).

This technical memorandum summarizes development plans and information used in preparing the prime contract statement of work specification for the DRDB major project ISSMM. It contains general information and philosophy for ISSMM and describes existing DREA capabilities under the general headings of development environment, Halifax Class geometric, structural and material databases, operational sea loads, global/detail finite element analysis, damage modelling and analysis, ultimate strength assessment, fatigue crack initiation and growth assessment, component stress and stability assessment, and mission risk assessment. This document is intended to provide optional guidance to the contract bidders in preparing their proposals. It also serves as a discussion of the state of the art in DND for ship structural analysis.

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus. e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified. If it not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title).

Ship structures
Finite element analysis
Maintenance
Sea loads
Fatigue
Ultimate strength
Object-oriented programming